



Atomic Energy  
Control Board

P.O. Box 1046  
Ottawa, Canada  
K1P 5S9

Commission de contrôle  
de l'énergie atomique

C.P. 1046  
Ottawa, Canada  
K1P 5S9

INFO-0315

PLANT AIR QUALITY FOLLOWING  
ACCIDENTAL RADIOACTIVE RELEASES

LITERATURE SURVEY  
AND MODEL TESTS ON  
RECIRCULATION AND INTERFERENCE  
FOR TWO IN-LINE CUBES

by

P.R. Slawson  
and  
G.J. Hitchman  
Envirodyne Limited

A research report prepared for the  
Atomic Energy Control Board  
Ottawa, Canada

Project Number 2.110.1

July 15, 1987



**PLANT AIR QUALITY FOLLOWING  
ACCIDENTAL RADIOACTIVE RELEASES**

This work was produced by Envirodyne Limited under contract to the Atomic Energy Control Board; contract DSS File No. 52SS-87035-6-4082.

---

**ABSTRACT**

Radionuclides exhausted to the atmosphere through vent stacks may contaminate the immediate building structure and/or nearby adjacent buildings resulting in hazardous contaminants entering the intake air. The objective of the work reported here was to obtain an overview of the analytical and experimental tools available to evaluate the transport and diffusion of vented effluents, and the nature of flows about structures, in order to predict the concentrations adjacent to and subsequently inside the immediate and nearby structures.

A scaling analysis established the requirements for physical modelling of flows about structures in wind and/or water tunnels. Subsequently, a series of test model studies were conducted in a water tunnel in the Mechanical Engineering Department of the University of Waterloo. This series of tests clearly indicates the effects that vent stack height and structure separation have on the recirculation and interference of exhaust vent contaminants for two model cube structures. The expected result that increased stack height and structure separation have on reducing recirculation and interference is clearly illustrated. Interference was still significant on the downstream structure when the vent stack height was increased to its maximum of one model cube height above the cube.

A literature survey was conducted to establish the nature and mathematical modelling of flows about structures. Also, mathematical modelling of the concentration field from roof-top vent effluents and subsequently the models for predicting concentrations within structures was investigated. The nature of flow about structures is very complex and not surprisingly the mathematical model predictions of concentrations about structures are subject to a wide error bound when compared with observed data. No specific data on concentrations within structures was found. The mathematical models are based on simple mass balances and are semi-empirical in nature. There is a strong need for well documented test on the concentration fields about model structures, with which more generic models may be developed. Very little data is available on multiple structures.

It is recommended that physical model studies be conducted, that simulate realistic exhaust and intake air vents with measurement of both the inside and the outside concentration fields. From this type of data one may test and/or further develop models that predict both inside and outside contamination.

**RÉSUMÉ**

Les radionucléides rejetés dans l'atmosphère à travers les cheminées de ventilation peuvent contaminer le bâtiment immédiat ou les bâtiments adjacents, ou les deux, entraînant ainsi la présence de contaminants dangereux dans l'air aspiré. Le présent travail avait pour but de tracer un aperçu des outils analytiques et expérimentaux disponibles pour évaluer le transport

et la diffusion des effluents à l'air libre, ainsi que la nature de débit d'air autour des structures, pour être en mesure de prévoir les concentrations à proximité et à l'intérieur des structures immédiates et adjacentes.

Une analyse de mise à l'échelle a établi les exigences relatives à la modélisation physique de débit d'air autour des structures dans des tunnels aérodynamiques ou hydrauliques, ou les deux. Par la suite, une série d'études de modèles a été effectuée dans le tunnel hydraulique du Département de génie mécanique de l'Université Waterloo où l'on a pu montrer clairement, dans le cas de deux modèles de cubes, les effets que la hauteur de la cheminée de ventilation et les séparations de la structure ont sur la recirculation et l'interférence des contaminants dans les sorties d'air. Les résultats prévus que des cheminées de ventilation plus hautes et de plus nombreuses séparations de la structure réduisent la recirculation et l'interférence sont clairement illustrés. L'interférence était toujours importante sur la structure en aval lorsque la hauteur de la cheminée de ventilation était élevée à son maximum d'une hauteur de cube modèle au-dessus du cube.

On a examiné la documentation pour établir la nature et la modélisation mathématique du débit d'air autour des structures. On a également examiné la modélisation mathématique du champ de concentration provenant des effluents des événements sur le toit, puis les modèles pour prévoir les concentrations à l'intérieur des structures. La nature du débit d'air autour des structures est très complexe et il ne faut pas s'étonner que les prédictions par modèles mathématiques du débit d'air autour des structures soient sujettes à une grande marge d'erreur en comparaison avec les données observées. On n'a découvert aucune donnée spécifique sur les concentrations à l'intérieur des structures. Les modèles mathématiques se basent sur de simples bilans massiques et sont de nature semi-empirique. Des données d'épreuves bien documentées sont extrêmement nécessaires sur les champs de concentration autour des structures modèles, à partir desquelles il serait possible de mettre au point plus de modèles génériques. Très peu de données sont disponibles sur les structures multiples.

On recommande d'effectuer des études de modèles physiques qui simulent des prises et des sorties d'air réalistes et de mesurer les champs de concentration autant à l'intérieur qu'à l'extérieur. A partir de ces données, il serait possible d'éprouver et de perfectionner des modèles de prévision pour la contamination interne et externe.

#### **DISCLAIMER**

The Atomic Energy Control Board is not responsible for the accuracy of the statements made or opinions expressed in this publication and neither the Board nor the authors assume liability with respect to any damage or loss incurred as a result of the use made of the information contained in this publication.

## Contents

|                                                                                                                              |           |
|------------------------------------------------------------------------------------------------------------------------------|-----------|
| <b>1 INTRODUCTION</b>                                                                                                        | <b>1</b>  |
| <b>2 MODELLING REQUIREMENTS AND SCALING ANALYSIS</b>                                                                         | <b>2</b>  |
| 2.1 Scaling Analysis.....                                                                                                    | 2         |
| 2.1.1 Similitude Requirements for the Planetary Boundary Layer<br>Based on the Governing Equations of Motion.....            | 2         |
| 2.1.2 Similitude Requirements Based on a Dimensional Analysis<br>Approach .....                                              | 5         |
| 2.2 Concluding Remarks on Modeling of Flows in the Planetary<br>Boundary Layer.....                                          | 7         |
| <b>3 FLOW ABOUT STRUCTURES</b>                                                                                               | <b>8</b>  |
| 3.1 Wakes Behind Structures.....                                                                                             | 8         |
| 3.2 Three-Dimensional Sharp Edged Obstacles (Cubes).....                                                                     | 8         |
| 3.2.1 Frontal Vortex Zone or Upwind Displacement Zone.....                                                                   | 8         |
| 3.3 The Effect of Ambient Turbulence on Wakes.....                                                                           | 13        |
| 3.4 Concentration Calculations On and Near Structures.....                                                                   | 14        |
| 3.4.1 Calculations of Concentration on the Roof of a Single Structure<br>Due to Sources On or Above the Roof (Region I)..... | 15        |
| 3.4.2 Concentration Calculations in the Near Wake (Region II and III of<br>Figure 5).....                                    | 19        |
| 3.4.3 Concentration Calculations in the Far Wake (Region IV of Figure 5)...                                                  | 19        |
| 3.5 Conclusions on Calculations of Building Wake Concentration Fields.....                                                   | 20        |
| 3.6 Concentration Calculations Inside Structures.....                                                                        | 20        |
| <b>4 EXPERIMENTAL DESIGN AND PROCEDURE</b>                                                                                   | <b>21</b> |
| 4.1 Modelling the Neutral Planetary Boundary Layer in a Water Channel.....                                                   | 22        |
| 4.2 Model Cubes.....                                                                                                         | 22        |
| 4.3 Model Stack Design.....                                                                                                  | 22        |
| <b>5 TEST MODEL EXPERIMENTS</b>                                                                                              | <b>24</b> |
| 5.1 Data Reduction and Analysis.....                                                                                         | 24        |
| 5.2 Results and Discussion.....                                                                                              | 24        |
| 5.3 Conclusions from Test Study.....                                                                                         | 37        |
| 5.3.1 Stack Height $0D$ (Flush-mounted Release).....                                                                         | 37        |
| 5.3.2 Stack Height $D/2$ .....                                                                                               | 37        |
| 5.3.3 Stack Height $D$ .....                                                                                                 | 37        |
| <b>6 CONCLUSIONS</b>                                                                                                         | <b>40</b> |
| 6.1 Model Test Study.....                                                                                                    | 40        |
| <b>7 RECOMMENDATIONS</b>                                                                                                     | <b>40</b> |
| <b>REFERENCES</b>                                                                                                            | <b>42</b> |



# 1 INTRODUCTION

The Atomic Energy Control Board is concerned with the potential contamination of site and indoor habitat due to vented radionuclides. Short exhaust vent stacks relative to adjacent building heights and/or low vent emission velocities and temperatures often result in the vented effluent being caught in the aerodynamic wake of the building from which the exhaust is vented. The exhaust plume may then fumigate sections of that building surface and potentially contaminate intake air (recirculation). As well, once the exhaust plume is entrapped in the immediate building wake, it may also contaminate intake air of any structures or buildings downwind (interference). Even if the building exhaust vent stack is high enough to escape the immediate building wake, (as a rule of thumb, 2-1/2 times the adjacent building height) it or portions of it, may still diffuse into the wake of nearby structures and interfere with the quality of that building's intake air.

The degree to which recirculation or interference occurs depends on ambient atmospheric conditions, exhaust vent source conditions and plant site topography and layout. Ambient atmospheric parameters of importance to recirculation and interference are windspeed, direction, and atmospheric stability. Clearly, the stronger the wind the greater the potential for any exhaust plume to be caught in the subsequent building wakes. Wind direction may be important depending on local topographical features adjacent to the plant site (ridges or valleys for example) and the geometry (aspect ratio) of buildings on site. It is well known that the intensity of building wakes is greatly influenced by their geometric aspect ratio and the wind direction relative to the long axis of a rectangular building. Atmospheric stability affects atmospheric turbulence levels which in turn affect the behaviour and growth of turbulent wakes and plumes, particularly at large distances from the source and/or structure.

It is clear that to simulate the recirculation interference potential for a given real plant site, one must be able to simulate the wind and turbulence structure as well as the source and geometry of the site. This was clearly demonstrated by Jain and Kennedy (1980) in a joint project with Slawson (1977) where field studies on cooling tower plume recirculation and interference were conducted at Plant Gaston in Alabama and later simulated in the IOWA Institute of Hydraulic Research water tunnel. This work involved simulations that were restricted to so-called near-field plume and wake behavior.

In a neutral or adiabatic atmosphere which is generally associated with much stronger winds than that found in unstable and stable atmospheres, the turbulence structure is dominated by mechanically or shear induced turbulence, and there is no net force exerted by the atmosphere on an exhaust plume restricting or enhancing its rate of rise. The potential for greatest aerodynamic wake-plume interaction in the near-field will occur in a neutral atmosphere, and is thus the atmospheric condition that one best first simulates in any model study.

The overall objective of the work reported here was to obtain an overview of the analytical, numerical and experimental tools available to evaluate the *near-field* (zone of aerodynamic influences of source geometry and initial conditions) behavior of the flows (plumes and wakes) about structures which can be used to predict quantitatively, the recirculation, interference and subsequently the ingress of previously discharged contaminants.

## 2 MODELLING REQUIREMENTS AND SCALING ANALYSIS

Accurate simulation of a flow in a laboratory model requires that certain dimensionless parameters describing the geometric, kinematic, and dynamic properties of the flow and its boundaries assume equal values in model and prototype. These dimensionless parameters may be derived by A: writing the governing equations relevant to the phenomena acting in the model and prototype in dimensionless form, or by B: dimensional analysis coupled with the insight into the nature of the physical processes involved. Since it generally is not possible to reproduce in a physical model all features of the flow under investigation, only the most important ones are modelled. The absence of complete similitude among the processes of secondary importance produces so-called model scale effects. If the model is to reproduce prototype behavior faithfully, the modelling must be done so as to minimize scale effects and to limit them to aspects of the phenomenon under study which do not have a major influence on the flow.

### 2.1 Scaling Analysis

#### 2.1.1 Similitude Requirements for the Planetary Boundary Layer Based on the Governing Equations of Motion

Similarity criteria for the application of air and water models to predict the dispersion of pollutants in the atmosphere have been reviewed by Snyder (1972) and Riley and Delisi (1977). Snyder derived the dimensionless parameters by nondimensionalizing the equations governing the instantaneous flow field (momentum, continuity, energy, and mass diffusion). The turbulence structure was treated as an initial or boundary condition on the instantaneous flow, which Snyder (1972) showed could be simulated through attention to appropriate turbulence scales.

We can write the governing equations for the planetary boundary layer after Panofsky and Dutton (1984) as:

Conservation of momentum:

$$\frac{\partial u_i}{\partial t} + u_k \frac{\partial u_i}{\partial x_k} = -\frac{1}{\rho_0} \frac{\partial p'}{\partial x_i} + g \frac{\theta'}{\theta_0} \delta_{i3} + \nu \frac{\partial^2 u_i}{\partial x_k^2} - 2\epsilon_{ijk} u_k \Omega_j \quad (1)$$

Conservation of mass:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

Conservation of energy:

$$\frac{\partial \theta'}{\partial t} + u_k \frac{\partial \theta'}{\partial x_k} = K \frac{\partial^2 \theta'}{\partial x_k^2} \quad (3)$$

Equations (1), (2), and (3) are the Boussinesq equations and are valid for incompressible flow in the absence of significant heating by radiation and or condensation. As such, they apply to a near-neutral atmosphere. In the above equations,  $u_i$  is the instantaneous velocity,  $\theta'$  and  $p'$  are small deviations in temperature and pressure from those in a neutral atmosphere ( $\theta_0, \rho_0$ ),  $\nu$  is the kinematic viscosity of the fluid and  $K$  is thermal diffusivity,  $\epsilon_{ijk}$  is the alternating tensor and  $\delta_{ij}$  is the Kronecker delta. The assumptions inherent in equations (1), (2) and (3) are:

- (1) incompressible flow
- (2) atmosphere composed of a perfect gas
- (3) deviations in pressure, temperature and density are small compared to the reference state of a neutral atmosphere

- (4) variations in  $\nu$  and  $K$  are negligible
- (5) no heat generated through viscous effects
- (6) no sources of any kind.

We next define our reference scales of length, velocity, temperature and angular rotation of the earth as:  $L, U_R, \theta_R, \Omega_R$ . These now allow us to non-dimensionalize our flow variables. Thus, we define non-dimensional variables as:

$$\begin{aligned} X_i &= \frac{x_i}{L} & U_i &= \frac{u_i}{U_R} \\ T &= \frac{U_R t}{L} & \rho' &= \frac{\rho}{\rho_R} \\ P &= \frac{p'}{\rho_R U_R^2} & \Theta &= \frac{\theta'}{\theta_R} \\ \Omega'_j &= \frac{\Omega_j}{\Omega_R} \end{aligned}$$

Substituting these into equations (1), (2) and (3) and simplifying results in:

$$\frac{\partial U_i}{\partial T} + U_j \frac{\partial U_i}{\partial X_j} + \frac{2}{Ro} \epsilon_{ijk} u_k \Omega'_j = -\frac{1}{\rho'} \frac{\partial P}{\partial X_i} + \frac{\Theta}{Fr^2} \delta_{3i} + \frac{1}{Re} \frac{\partial^2 U_i}{\partial X_j^2} \quad (4)$$

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (5)$$

$$\frac{\partial \Theta}{\partial T} + U_i \frac{\partial \Theta}{\partial X_i} = \frac{1}{Pe} \frac{\partial^2 \Theta}{\partial X_i^2} \quad (6)$$

|       |                                               |                                   |
|-------|-----------------------------------------------|-----------------------------------|
| where | $Ro \equiv U_R / \Omega_R L$                  | is a Rossby number,               |
|       | $Fr \equiv U_R / (gL\theta_R/\theta_0)^{1/2}$ | is the densimetric Froude number, |
|       | $Re \equiv U_R L / \nu$                       | is the Reynolds number,           |
| and   | $Pe \equiv U_R L / K$                         | is the Peclet number.             |

Equations (4), (5), and (6) comprise five equations in five unknowns (three velocity components, pressure and temperature) and as such are in principle solvable. These equations are applicable to the instantaneous flow field and as such, apply to both laminar and turbulent flow. Thus, any two flows will be similar if they are governed by identical equations (4) (5) and (6). The solution sets to equations (4) (5) and (6) will be identical if the coefficients  $Ro, Fr, Re$ , and  $Pe$  along with the non-dimensionalized boundary conditions are also identical.

Introduction of the molecular diffusion equation which is identical in form to the energy equation (3), allows us to simulate a passive contaminant within the flow field. The molecular diffusion equation is given by:

$$\frac{\partial c}{\partial t} + u_i \frac{\partial c}{\partial x_i} = \alpha \frac{\partial^2 c}{\partial x_i^2} \quad (7)$$

or in non-dimensional form

$$\frac{\partial C}{\partial T} + U_i \frac{\partial C}{\partial X_i} = \frac{1}{ReSc} \frac{\partial^2 C}{\partial X_i^2} \quad (8)$$

where  $Sc \equiv \nu/\alpha$  is the Schmidt number and  $\alpha$  is a mass diffusivity.

It would appear then that we must maintain identical non-dimensional parameters,  $Ro$ ,  $Fr$ ,  $Re$ ,  $Pe$  and  $Sc$  in order to accurately simulate an atmospheric flow in a water or wind tunnel. In addition we must maintain geometric similarity and similarity of boundary conditions.

Let's now consider the specific nature of the flow we wish to simulate. We are interested in the near-field behavior of plume and wake flows adjacent to or near structures. Pasquill (1962), Corrsin (1953), Saffman (1962), Howroyd and Slawson (1978) have shown that within 5 km of the source in a neutral atmosphere, Coriolis forces have an insignificant effect on the flow field and dispersion process. Thus, for our purposes, we can ignore  $Ro$  similitude as we are interested in typically ten building heights downstream or of order 300 m.

Since we are to model a neutral boundary layer, the ambient densimetric Froude number ( $Fr$ ) is also unimportant as a scaling criteria. We note that we can re-write  $Pe$  as:

$$Pe = \frac{U_R L}{K} = \frac{U_R L}{\nu} \cdot \frac{\nu}{K} = Re \cdot Pr$$

and that

$$Re \cdot Sc = \frac{U_R L}{\nu} \cdot \frac{\nu}{\alpha}$$

Thus, both these terms are products of a Reynolds number and the ratio of molecular properties of the fluid (not flow properties). Since we are interested in turbulent transport the  $Pe$  and  $Sc$  numbers are unimportant to our modelling.

We are thus left with requiring similarity in Reynolds number alone. Golden (1961), Smith (1951), Halitsky (1963) have shown that one would obtain Reynolds number independence in the modelled flow field if the  $Re$  were high enough.

These authors suggest that a  $Re$  of  $2 \times 10^4$  up to  $7.9 \times 10^4$  may be required for Reynolds number independence for flows about cubes to flows about round topped structures. This suggests that we might approach Reynolds number independence for our flow about our cubic structures at a  $Re \approx 10^4$ .

Ungate et al (1975) have shown that the modelling of turbulent round jets is acceptable for jet Reynolds numbers of greater than a critical level of 1500. Jain and Kennedy (1979) suggest that modelling of the ambient turbulence is not critical to successful modelling of the very near field behavior of the wake behind a structure.

The Reynolds number is the ratio of inertial forces to viscous forces. The scale reductions commonly employed in model studies result in model Reynolds numbers that are typically three or four orders of magnitude smaller than those in the prototype. For turbulent flows it is not necessary to reproduce prototype Reynolds numbers exactly in the model, provided that both the ambient and jet flows are fully turbulent. The justification for using smaller Reynolds numbers in a model is based on the hypothesis put forth by Townsend (1956) that the large-scale features of the flow are dynamically similar at all sufficiently high Reynolds numbers. It is these large scale features that are dominant in turbulent mixing. This may be explained further by considering the turbulent stress (Reynolds stresses) terms that arise in the time-averaged equations of motion for turbulent flow. For example,

$$\overline{u'w'} = -K_M \frac{\partial \bar{u}}{\partial z}$$

where  $\overline{u'w'}$  is the time averaged product of instantaneous fluctuating velocities and is modelled using the exchange coefficient hypothesis and  $K_M$  is a turbulent eddy exchange coefficient for momentum.  $K_M$  may be thought of as composed of the product of a turbulent length scale and turbulent eddy velocity scale ( $l_t, U_t$ ).

Thus,  $K_M \propto l_t U_t$  and in the surface layer of a neutral planetary boundary layer  $K_M \propto U_* Z_0$ , where  $U_*$  is a friction velocity and  $Z_0$  a roughness element scale. Thus we see that  $l_t$  scales with  $Z_0$ , and  $U_t$  scales with  $U_*$ . Also,  $U/U_*$  scales with  $Z/Z_0$  and therefore the only new scale introduced is the roughness scale  $Z_0$ . We conclude that similarity of the large scale features of turbulence ( $l_t$ ) may be obtained by correctly scaling the roughness elements in model to that of the prototype in a neutral boundary layer.

Satisfactory modelling of the planetary boundary layer may be achieved following the work of Counihan (1969), (1975), Counihan et al (1974), and Cheah et al (1983). These techniques consist of constructing vortex generators of wedge shape in plan view and quarter ellipses in elevation.

We conclude from our scale analysis based on the governing equations of motion that: (i) only Reynolds number similarity must be maintained in model and prototype for the simulation of the neutral planetary boundary layer, (ii) exact similarity of Reynolds number is not required if the flow in the model is made Reynolds number independent (of order  $10^4 - 10^6$ ), and (iii) appropriate simulation of turbulence may be achieved through the use of vortex generators, surface roughness modifications and trip fences after Counihan (1969).

### 2.1.2 Similitude Requirements Based On a Dimensional Analysis Approach

The independent parameters of interest in the study of a buoyant or passive stack release above a cubic structure in a neutral planetary boundary layer are: stack height =  $h_s$ , stack diameter =  $d$ , building height =  $H$ , building spacing =  $S$ , exit velocity =  $V_j$ , stack effluent density =  $\rho_j$ , ambient density =  $\rho_a$ , ambient free stream velocity =  $U_a$ , atmospheric boundary layer thickness =  $\delta$ , representative roughness scale  $Z_0$ , kinematic viscosity of the effluent =  $\nu_j$ , kinematic viscosity of ambient fluid =  $\nu_a$ , molecular thermal diffusivity of the effluent =  $K_j$ , molecular mass diffusivity of the effluent =  $\alpha_j$ , gravitational constant =  $g$ , difference between the ambient and adiabatic lapse rates =  $\Delta\theta$ , and the angular rotation of the earth =  $\Omega$ . The dependent quantity of primary concern is the spatial distribution of the concentration of the stack effluent, or in our case  $\Delta T/\Delta T_0$ , where  $\Delta T = T - T_a$ ,  $\Delta T_0 = T_j - T_a$ , and  $T$  is the temperature at any point in the fluid. Subscript  $a$  refers to ambient while subscript  $j$  refers to a stack effluent variable.

It is worthy to note here that Golden (1961) has shown that  $\Delta T/\Delta T_0$  may be used as a representative passive tracer for values of a Froude number ( $Fr_a$ ) based on ambient velocity and structure length scale ( $L_s$ ) of above 0.8 for studies of the interaction of plumes and wakes.

We can now write our dependent variable  $\Delta T/\Delta T_0$  as a function of a group of non-dimensional variables viz

$$\frac{\Delta T}{\Delta T_0} = f\left(\frac{h_s}{d}, \frac{H}{d}, \frac{S}{d}, \frac{\delta}{d}, \frac{Z_0 V_j}{d U_a}, Fr_j, Re_j, Re_s, Ro, Fr_a, Pr, Sc\right) \quad (9)$$

where the Boussinesq approximation (neglect of density variation except in the buoyancy term) has been applied, and

$$Fr_j = \frac{V_j}{((\Delta\rho_s/\rho_a)gd)^{1/2}} = \text{jet densimetric Froude number}$$

$$\begin{aligned}
 Ro &= \frac{U_a}{\Omega d} = \text{Rossby number} \\
 R &= \frac{V_j}{U_a} = \text{jet exit velocity ratio} \\
 Fr_a &= \frac{U_a}{(gd(\Delta\theta/\theta))^{1/2}} = \text{ambient Froude number} \\
 Re_j &= \frac{V_j d}{\nu_j} = \text{jet Reynolds number} \\
 Pr &= \frac{\nu_j}{K_j} = \text{jet Prandtl number} \\
 Re_a &= \frac{U_a \delta}{\nu_a} = \text{ambient Reynolds number and} \\
 Sc &= \frac{\nu_j}{\alpha_j} = \text{Schmidt number}
 \end{aligned}$$

Here as in the scale analysis from the governing equations we can with similar reasoning, reduce our non-dimensional parameters to:  $Fr_j$ ,  $R$ ,  $Re_a$ . Of these, the  $Fr_j$  and  $R$  must be reproduced in model and in prototype if one is concerned with accurate simulation of the buoyant and momentum rise of the stack effluent. In our present study, since we are not attempting to simulate a particular source, we can relax this requirement. Also, for  $R$  values less than 1.5, stack downwash will become increasingly intense.

Again, provided we are interested only in the very near-field flow about our cubic models only a suitably high enough Reynolds number must be maintained in our modelled flow.

Considering our wake turbulence, we can represent this turbulent flow field by a suitable eddy exchange coefficient for the wake say  $K_w$ , where  $K_w \propto l_w V_w$  where  $l_w$  and  $V_w$  are turbulent eddy length and velocity scales for wake diffusion. The appropriate eddy length scale in a wake must scale with the dimension  $H$  and the wake eddy velocity scale must scale with the ambient velocity scale  $U_a$ . Thus, similarity of wake flows should be maintained provided the model structure dimension (cube) is also scaled.

From equation (9), we see that our non-dimensional length scales may be re-written in the form  $h_s/H$ ,  $\delta/H$ ,  $S/H$ ,  $d/H$ ,  $Z_o/H$  where structure height become the important normalizing length scale in wake flows instead of plume or jet exit diameter  $d$ , for simulation of jets and plumes.

The above non-dimensional length scales are generally, relatively easy to satisfy. However,  $\delta/H$ , the ratio of the planetary boundary layer height to the building height requires that the planetary boundary layer be simulated and the constant shear stress layer or surface boundary layer be established. Counihan, Hunt and Jackson (1974), used a  $\delta/H = 8$ , and also pointed out that  $Z_o \ll H \ll \delta$ .

In summary, for modelling plumes, dynamic similarity is insured when the values of the jet densimetric Froude number,  $Fr_j$ , and the velocity ratio,  $R$ , are equal in model and prototype, provided  $Re_j$  is above the critical level of 1500. Equating the jet densimetric Froude numbers for the model and prototype yields

$$V_r = \sqrt{(\Delta\rho_o/\rho_a)_r d_r} \quad (10)$$

where the subscript  $r$  denotes the ratio between any two corresponding quantities in the model and the prototype. Limitations of laboratory facilities generally impose restrictions on each of the three ratios appearing in (10). In principle, a laboratory model study could be

conducted using either air or water. However, for reasonable values of  $d_r$  and  $(\Delta\rho_s/\rho_s)_r$ , for both water and air, the model has to operate at quite low velocities. For example, a typical prototype value of  $(\Delta\rho_s/\rho_s)$  is 0.04, and a convenient model scale is  $d_r = 1/150$ . Practical considerations (including heating requirements) limit the temperature rises of laboratory water and air to about  $10^\circ\text{F}$  ( $5.6^\circ\text{C}$ ) and  $60^\circ\text{F}$  ( $33.3^\circ\text{C}$ ), respectively. The resulting velocity ratios given by (10) are  $V_r = 0.0161$  for water and  $V_r = 0.130$  for air. For a prototype stack effluent velocity of 30 fps (9.1 m/s), the corresponding model velocities are 0.48 fps (15 cm/s) for water and 3.90 fps (1.2 m/s) for air. Experiments using water with velocities of the order of a few tenths of a meter per second pose no difficulties in measuring the velocities, corresponding discharges, etc., while air experiments with velocities of a meter per second are considerably more difficult. Indeed, it is practically impossible to obtain a suitably distributed air cross-flow of this magnitude, as even a low value of  $R$ , say  $R = 1$ , would require. To obtain model values of  $(\Delta\rho_s/\rho_s)$  that are sufficiently large to give acceptably large model values of  $U_s$  in wind tunnels, use has been made of low-density gases. Plume mapping then was done by point sampling and subsequent chemical analysis to determine the concentration of the density-biasing gas and thus of stack effluent. Needless to say, this is far more difficult than measurement of temperature in place, as can be done in water flume experiments.

## 2.2 Concluding Remarks on Modeling of Flows in the Planetary Boundary Layer

(1) For the simulation of a neutral planetary boundary layer, only a sufficiently high enough ambient Reynolds number must be maintained, that is:

$$Re_s = \frac{U_s \delta}{\nu} > 10^4 \quad \text{for Reynolds number independence.}$$

(2) Every effort should be made to model predominantly the large scale structure of turbulence. This may be accomplished using the techniques of Counihan (1969), Cheah et al (1983). Reviews by Davenport and Isyumov (1967), Cermak and Arya (1970) and Hunt and Fernholz (1975) show the necessity of correctly modelling the planetary boundary layer to assess wind loads on buildings, flow around buildings and atmospheric dispersion. Jain and Kennedy (1980) suggest that the correct modelling of the turbulence structure is not important for the simulation of very near-field wake flows. It would appear that some effort should be made to determine the effect ambient turbulence has on a given test or model set-up.

(3) To minimize surface viscous effects in the model, Snyder (1972), suggest that the approach flow surface Reynolds number

$$Re_{so} = \frac{U_s Z_o}{\nu} \geq 2.5$$

to ensure an aerodynamically rough surface. He also advises that the surfaces of the models (in our case cubes) be roughened. Vet (1978) suggests stipeltone paint. The object is to have the roughness elements on the model surface extend out of the viscous sub-layer so that form drag becomes predominant in producing shear inducing turbulence.

(4) The model must not block the oncoming flow. Blockage effects are considered minimal if the plume/wake cross sectional area normal to the approach flow is  $< 50\%$  of the working cross-section of the water or wind tunnel (Bugler and Tatincloux (1974), Hoult and Weil (1972)).

(5) The controlling non-dimensional parameters for modelling heated jets in a cross-flow are the jet densimetric Froude number ( $Fr_j$ ) and the jet exit velocity to wind speed ratio ( $R$ ).

(6) It is not necessary to model Coriolis' effects (Rossby number) when simulating a neutral planetary boundary layer and dispersion within 5 km of a source.

(7) Temperature may be used as a very convenient passive tracer in water channel simulations of jet and building wake interaction provided the densimetric Froude number defined as  $Fr_\rho = U_\infty / ((\Delta\rho/\rho_\infty)gL_s)^{1/2}$  is above 0.8.

(8) Yang and Meroney (1970), have shown that atmospheric stability has insignificant effect on the flow field and dispersion near buildings, within 5 building heights of the source. Hatcher et al (1978) reported similar findings in a study of flow about a reactor complex.

### 3 FLOW ABOUT STRUCTURES

#### 3.1 Wakes Behind Structures

The effluent released from a vent stack on a building may escape the wake of the structure entirely, or be partially or completely trapped within the wake. In order to assess stack plume building wake interaction, we must understand both the plume and building wake dynamics. This section of our literature review focuses on building wakes, their growth and dynamic structure.

The flow about three dimensional structures is extremely complex and deals with the most intractable aspects of fluid mechanics, that of complex turbulent shear flow.

Halitsky (1963a), (1963b), (1963) Slawson (1972), Meroney (1970) and many others, present various simple models for the prediction of pollutant concentrations in the wakes of buildings. A number of the models for estimating contaminant concentrations near buildings require a detailed knowledge of building wake structure. Such detailed information on building wakes is extremely limited. There have been a few studies conducted that attempt to characterize the gross features such as wake growth behind simple geometric shapes and much fewer that provide details of the wake turbulent structure itself. We will focus on those studies that provide information on wake characteristics behind three dimensional structures, with emphasis on the work on cubic structures that may support the experimental objectives of this present study.

#### 3.2 Three-Dimensional Sharp Edged Obstacles (Cubes)

##### 3.2.1 Frontal Vortex Zone or Upwind Displacement Zone

From Figures 1, 2, and 3 we see that there is a frontal vortex zone or zone where the upwind flow senses the presence of the cube as it approaches it. Current ability to predict the extent of this zone is very limited. For our purposes we may consider that this zone, although affecting the other regions of the wake is not of direct interest as we do not expect roof vent releases to diffuse into this region. However, we will summarize some of the expected general features of this zone.

As the upstream flow approaches the cube, it decelerates in the direction of the mean flow as it approaches the cube and accelerates both laterally and vertically to pass around the cube. This gross deflection of the incident flow strongly affects the shear-induced standing vortex on the windward side of the cube. The size and strength of this vortex depend on the curvature of the incident velocity profile relative to the height of the structure and on the width to height ratio.

Corke and Nagib (1976) determined from wind tunnel tests on a block of  $L/H - W/H = 0.52$  that the diameter of the standing vortex increase with increasing wind profile exponent. Their wind profile was of the form  $u \propto z^n$  where  $n$  ranged from 0.11 to 0.35. The dividing point between upward flow and downward flow (top of frontal vortex, see Figure 2) ranged

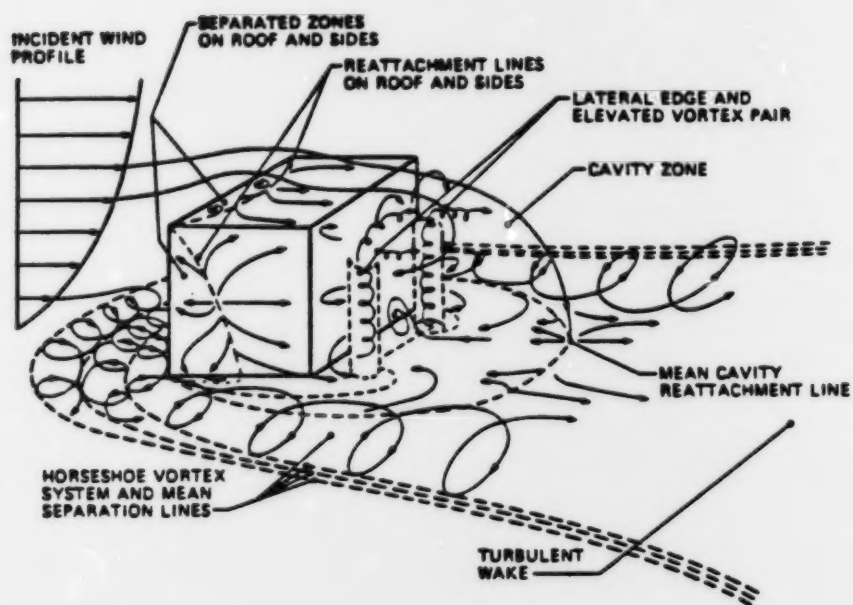


Figure 1: Model of Flow Near a Sharp-edged Three-dimensional Building in a Deep Boundary Layer (from Hosker (1984))

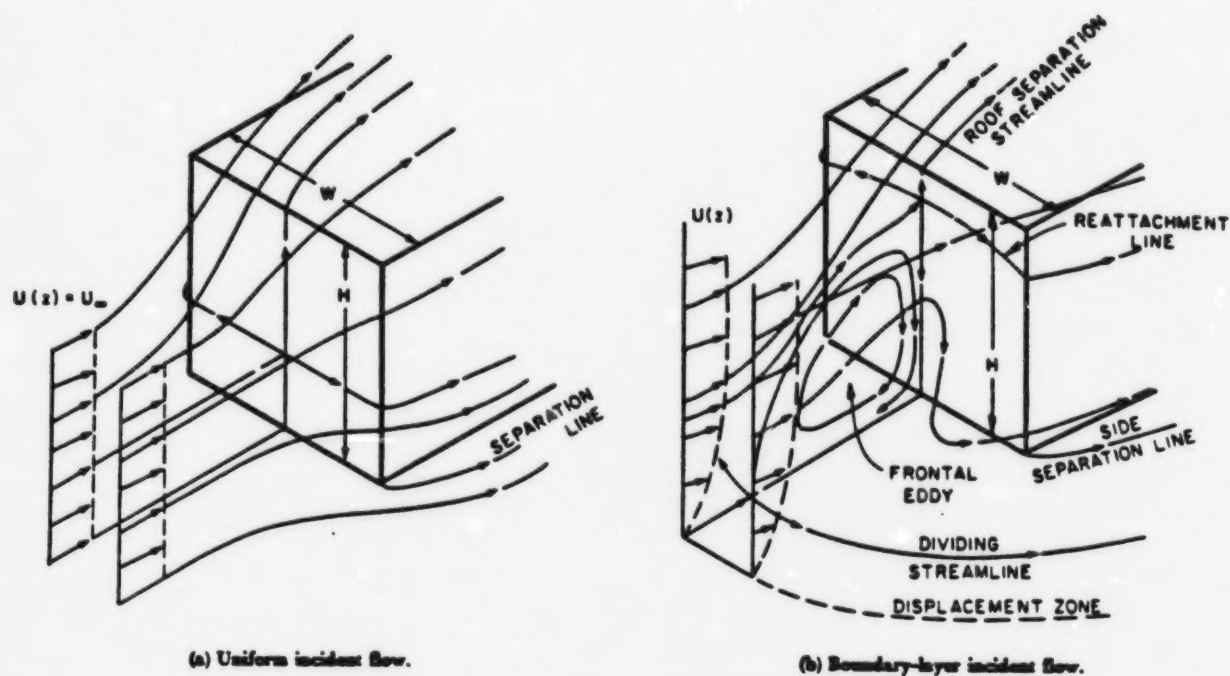


Figure 2: Flow Patterns Upwind of a Cube for Different Incident Flows (from Baines (1963))

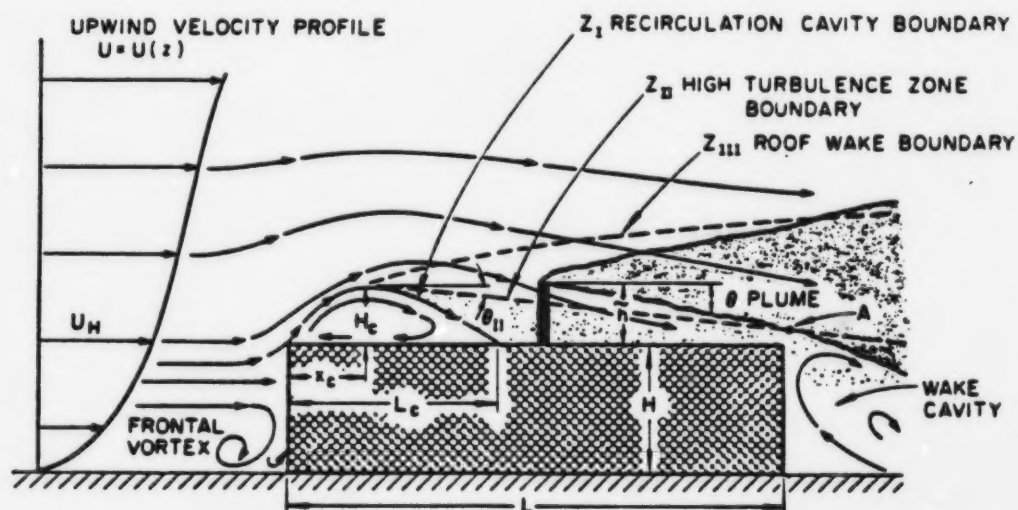


Figure 3: Flow Over Center of Long Flat Building Roof for Wind Perpendicular to Upwind Face (from Wilson (1979))

from  $z/H = .33$  to  $0.77$  for  $n = 0.11$  to  $0.33$  (see re-attachment line on block in Figure 2).

The upwind extent of this frontal vortex zone is where the flow separates from the ground to lift over and around the obstacle. This point is also not well documented in the literature. However, it is believed that it is unsteady as it depends on the shear in the mean flow, upstream turbulence and feedback flow perturbations such as vortex shedding off the structure into the wake (Norman 1972).

Near-wake flow as defined in Figures 1, 2, and 3 is extremely complicated and depends on the shape of the structure, its roughness, the nature of the incident flow as well as mean flow direction relative to the geometry of the structure. The result is a complicated separated turbulent shear flow with flow reversals and or standing vortices.

Referring to Figure 3, the region denoted by  $H_c$  is the separation bubble or zone defining the incident flow separation from the leading roof edge and subsequent re-attachment on the roof of the flat roofed structure. This separation zone or roof cavity will exist on flat roofed structures providing  $L/H$  is unity or larger (Hosker 1979).

Wilson (1979), working with various block structures found that the length of this roof cavity is given by

$$L_c \approx 0.9\bar{R}$$

$$\text{where } \bar{R} \equiv \zeta_1^{2/3} \zeta_2^{1/3},$$

and where  $\zeta_1$  is the smaller of  $H$  and  $W$ , and  $\zeta_2$  the larger. If this applies to a cubic structure, then

$$L_c \approx 0.9H.$$

Wilson's work was for the case where the incident flow was perpendicular to the width of the block structures.

In view of the uncertainties associated with the measurements and limitations of the data base, we can probably safely assume

$$L_c \approx H$$

for cubic structures. Wilson also found that the maximum height reached by the roof cavity is

$$H_c = 0.22\bar{R}$$

or again, extrapolating to cubic structures

$$H_c = 0.22H$$

and this occurs at a distance  $\bar{R}/2$  from the leading edge of the roof. Thus, our important recirculation zones may be described as illustrated in Figure 4 for a single isolated cubic structure.

Thus, Wilson's work suggests that the roof cavity region extends over the entire surface and re-attaches on the downstream edge of a cube. Hosker (1979) suggests that the wake cavity region ( $X_r$ ) or zone would extend to  $\sim 2H$  downstream of a cubic structure.

The upwind flow (separation zone) is typically  $H$  to  $1.3H$  in length after Frost and Shahabi (1977), Wilson and Netterville (1978), and its separation point is unsteady as noted earlier.

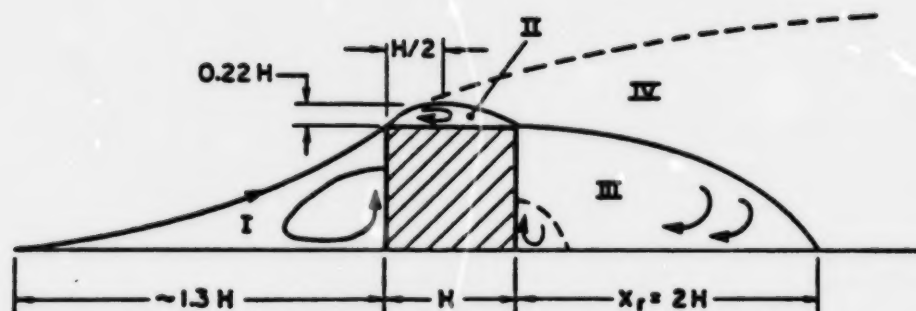


Figure 4: Separation Zones Around a Single Isolated Structure  
I - Frontal Separation Zone, II Roof Cavity, III - Rear Wake Cavity, IV - Far Wake

Referring once more to Figure 3, Wilson finds

$$\frac{Z_{II}}{\bar{R}} = 0.27 - 0.1 \frac{x}{\bar{R}}$$

and

$$\frac{Z_{III}}{\bar{R}} \approx 0.28 \left( \frac{x}{\bar{R}} \right)^{1/3}$$

Thus, for cubic structure where  $\bar{R} = H$  we get

$$\frac{Z_{III}}{H} = 0.27 - 0.1 \frac{x}{H}$$

and

$$\frac{Z_{III}}{H} = 0.28 \left( \frac{x}{H} \right)^{1/3} \text{ for } x > \frac{H}{2}$$

where  $x$  is measured from the upwind edge of the building. Hosker (1979), suggests the following equations for the wake cavity region II of Figure 4 behind flat roofed block structures for  $L/H \leq 2$  based on analysis of Evans (1957) data

$$\frac{X_r}{H} = \frac{L}{H} + \frac{A(W/H)}{1 + B(W/H)} \quad (11)$$

$$\text{where } A = -2.0 + 3.7 \left( \frac{L}{H} \right)^{-1/3}$$

$$\text{and } B = -0.15 + 0.305 \left( \frac{L}{H} \right)^{-1/3}$$

Thus for cubic structures like our models (11) becomes:

$$\frac{X_r}{H} = 2.3$$

Yang and Meroney (1970), find the wake cavity to be  $2H$  to  $3H$  behind a cubic model. Counihan (1971) on the other hand reports a wake cavity of only  $1$  to  $1.5H$  behind a cubic model. Thus, it would appear that the region denoted by  $X_r$  ranges from  $H$  to  $3H$ . In Meroney and Yang's (1970) work, the flow had re-attached to the top and sides of the models. Castro and Robins (1977) found that for sufficiently deep incident boundary layers, the flow always reattached to the top and sides of their cubic models and occurred at a distance  $H/4$  from the leading edge. They also (as Meroney and Yang) found that the end of the wake cavity was then about  $1.5H$  from the rear face of the model. Vincent (1978) found

$$X_r/H \approx 1.3 \text{ to } 1.8 \text{ for } \delta/H \approx 20 \text{ to } \delta/H = 8$$

where  $\delta$  is the depth of the incident boundary layer.

We conclude from the above work that for stand alone cubical objects, the extent of the wake cavity zone is still not well understood, and much further work is required.

### 3.3 The Effect of Ambient Turbulence on Wakes

Blessman (1971), Castro and Robins (1977) studied surface mounted three-dimensional objects (cubes) in both smooth and turbulent winds. The major effects of incident turbulence seem to be related to enhanced thickening of and mixing along shear layers shed from the windward surfaces of sharp edged objects promoting flow reattachment and resulting in smaller wake cavities. On rounded obstacles, flow separation is delayed. The net effect is akin to that produced by roughening the body surface or increasing the Reynold's number of the flow which, of course, are procedures widely used in wind and water tunnel testing.

Vet (1978), conducted flow visualization experiments in a water channel on several basic block structures including surface mounted cubes. He also looked at the effect of cube spacing on wake structure. For a single isolated cube, Vet found for  $0^\circ$  orientation:

$$\begin{aligned} \frac{R_s}{H} &= 1.22 \left( \frac{x}{H} \right)^{0.06} & \frac{x}{H} &\leq 0.27 \\ \frac{R_s}{H} &= 1.341 \left( \frac{x}{w} \right)^{0.14} & 0.27 < \frac{x}{H} < 24.1 \\ \frac{R_w}{H} &= 0.92 \left( \frac{x}{H} \right)^{0.18} & \frac{x}{H} &\leq 1.28 \\ \text{and } \frac{R_w}{H} &= 0.84 \left( \frac{x}{H} \right)^{0.8} & 1.28 < \frac{x}{H} < 25 \end{aligned}$$

where  $R_w$  and  $R_s$  are the half width and height of the wake behind the surface mounted cubes. Vet further shows that when using a simple concentration formulae given by Slawson (1972), that for a low wide rectangular building the normalized mean concentration is given by

$$\frac{\bar{z}}{Q} \propto x^{-0.60}$$

where  $Q$  is the initial concentration of the contaminant and for a cubic building

$$\frac{\bar{z}}{Q} \propto x^{-0.64}$$

Comparison with the field data of Dickson et al (1969) is remarkably good in that Dickson's data also indicates

$$\frac{\bar{z}}{Q} \propto x^{-0.60}$$

Yang and Merony (1970) indicate from wind tunnel measurements behind a cubic model that

$$\frac{\bar{z}}{Q} \propto x^{-0.6} \text{ to } x^{-0.7}$$

which compares favourably with that found by Vet for a model cube.

There is certainly a significant difference between Vets vertical growth of the wake behind a cube and that suggested by Wilson (1979), that is Vet suggests

$$\frac{R_w}{H} \propto \left(\frac{x}{H}\right)^{0.14}$$

while Wilson suggests

$$\frac{R_w}{H} \propto \left(\frac{x}{H}\right)^{0.33}$$

Vet finds

$$\frac{R_w}{H} \propto \left(\frac{x}{H}\right)^{0.28}$$

for flow over a two-dimensional structure. Hosker (1982) in his review calls for additional measurements describing the wake envelope. Thus, we conclude that there is still a great deal of uncertainty in the rates of growth of wakes behind even simple single block structures.

### 3.4 Concentration Calculations On and Near Structures

In order to calculate the concentrations on and near structures from various source locations one requires at the very least:

- (1) a knowledge of the *wake envelope*
- (2) a detailed knowledge of specialized flow regions within the wake envelope
- (3) a knowledge of source height, and emission characteristics
- (4) geometry of the source structure and adjacent structures.

Figure 5 illustrates four general regions of dynamical importance within the wake envelope about structures. The calculation of concentrations within the wake available in the literature appear to focus on regions I, II, and IV pertaining to a single structure with little theoretical work available for multiple structures.

I - on the roof itself.

II - in the immediate wake cavity region.

III - the cavity zone between adjacent structures.

IV - in the far-field wake region.

We are primarily interested in regions I, II, and III with only a minor interest in region IV for the present report and two cube geometry that we have selected.



Figure 5: Regions of Dynamical Importance Within the Wake Envelope about Structures

### 3.4.1 Calculations of Concentration on the Roof of a Single Structure Due to Sources On or Above the Roof - (Region I)

Figure 6 illustrates the expected plume-wake interaction behavior from a source located on top of a structure. One may have either partial interaction (a), complete interaction (b) and fumigation of the downstream face of the structure or no interaction of plume and wake (c).

#### Roof Top Flush-Mounted Sources

Wilson (1985) describes a semi-empirical model for calculating the dilution of exhaust gases from flush-mount roof-top vents on a single cubic structure. He obtains a formula for minimum dilution as a function of distance from the source. It is clear from Wilson's work that there is a great deal of empiricism built into his minimum dilution formula. In a test of his model, Wilson compares the model prediction with the full scale data of Sagendorf et al (1979). Differences between predicted and full scale concentrations are less than a factor of two for this particular study with the predicted concentrations exceeding those measured (assumed on the roof of the structure).

Wilson, points out that for roof-top calculations:

- (1) there is no significant effect of building size or shape on minimum dilution.
- (2) dilution increases in direct proportion to exhaust flow rate.
- (3) wind direction relative to building orientation is only important for low exhaust velocity to wind speed ratios.
- (4) no account of plume buoyancy has been adequately dealt with, which is very important at low wind speeds.
- (5) the first principle of good intake-exhaust design is to locate intakes in the lower 1/3 of buildings with exhausts on the roof.

Wilson's formula for minimum dilution takes the form found from a simple mass balance between the source and that at some distance  $s$  from the source.

$$D_{min} \propto \left[ A + \left( \frac{Bs^3}{RAe} \right)^{1/2} \right]^2 \quad (12)$$

where  $A$  is an initial empirical dilution factor,  $B$  accounts for the effects of averaging time

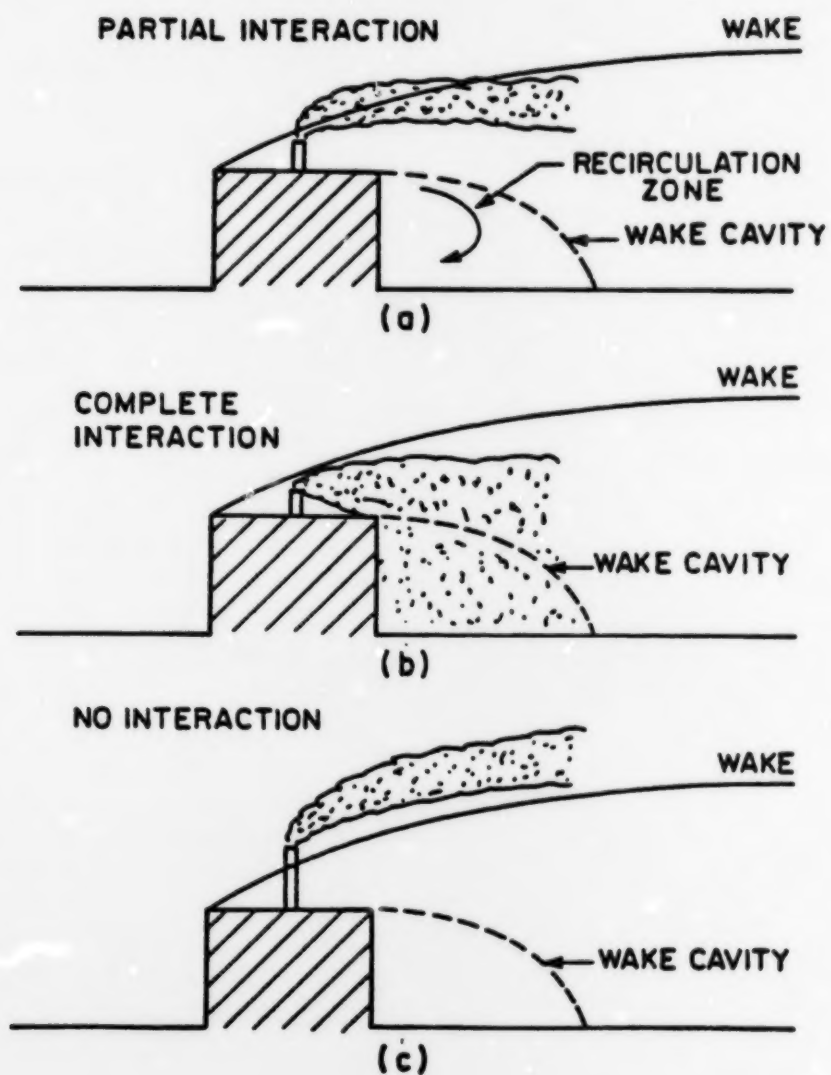


Figure 6: Plume-wake Interaction Behavior from a Roof-top Source

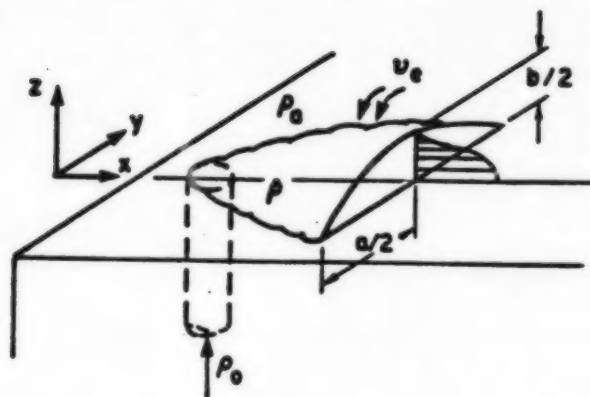


Figure 7: A Flush-Mounted Roof-top Source and Plume

on dilution (taken from Pasquill (1974),  $R$  is the exit velocity to windspeed ratio and  $A_e$  is the area of the exhaust duct at exit.

The use of Pasquill's formulation for the effect of averaging time on dilution in roof-top building wake turbulence is certainly open to question. Rather than attempt to document the details of a largely empirical formula such as (12), we will simply derive the general form that roof-top concentration formulas might take.

We will assume that the exhaust plume is immediately bent-over (strong wind situation) and attaches itself to the roof surface as shown in Figure 7. Also, the plume is non-buoyant and is advected downwind at the constant roof-top wind speed  $U$ . Considering a differential element of the plume at a distance  $x$  downwind from the source, we obtain a mass balance given by:

$$\frac{d\rho AU}{dx} = \rho_a C v_e \quad (13)$$

where  $A$  is the area of the plume cross-section at  $x$  measured from the roof vent,  $C$  is the perimeter of the plane cross-section at  $x$ ,  $\rho$  and  $\rho_a$  are the exhaust and ambient densities respectively, and  $v_e$  is an entrainment velocity. If we assume that the density of the exhaust gas is approximately equal to that of ambient air ( $\rho \approx \rho_a$ ), then equation (13) reduces to an expression for the volume expansion of exhaust gases by the entrainment of ambient air. In order to solve equation (13) we need an assumption regarding the mechanism of entrainment which would reduce the dependent variables to one (plume cross-sectional area). Since the wake turbulence in the roof-top boundary layer is assumed to be responsible for plume spreading (dilution), and that turbulence depends on wind speed for a given structure (assuming a wind normal to roof edge) we may assume that  $v_e = \beta U$ . Thus (13) reduces to:

$$\frac{dA}{dx} = \beta C \quad (14)$$

If the plane cross-section were circular in shape, (14) suggests ( $A = \pi r^2/2$  for a radius  $r$ ), that upon integration

$$r = r_0 + \beta x \quad (15)$$

where  $r_0$  is an initial radius and for simplicity may be assumed small compared to  $r$  at short distances from the source such that

$$r = \beta x \quad (16)$$

Thus (15) or (16) suggest that with our above assumptions, the plume grows linearly with distance from the source. This derived result is an *a priori* assumption in Wilson's (1985) work. One may show that the linear growth law (16) is really a consequence of fully developed turbulent diffusion after long diffusion time (Slawson and Csanady (1967)). There is certainly some question as to whether it is valid for the near-field turbulent roof-top wake zone of diffusion for our exhaust vent. Nonetheless, the linear growth law facilitates a simple derivation for an expression of exhaust dilution, which for practical purposes ultimately requires considerable experiment and empirical tuning.

We note that (14) may be solved, given any defined plume cross-sectional shape. Since observation of diffusion near a surface, indicates that the vertical and horizontal scales of turbulence are different (vertical < horizontal), an assumed elliptical cross-sectional shape appears most appropriate.

The half area and half circumference of an ellipse is given by

$$A = \frac{\pi ab}{2}, C \approx \frac{\pi}{\sqrt{2}}(a^2 + b^2)^{1/2} \quad (17)$$

where  $a$  and  $b$  are the major and minor axis. If we assume Gaussian or normal distributions of contaminant within the plume where  $\sigma_x$  and  $\sigma_y$  are the vertical (b-axis) and horizontal (a-axis) standard deviations, then  $a \approx 4.3\sigma_y$ ,  $b \approx 4.3\sigma_x$ . As is often measured, we can also assume  $\sigma_y = m\sigma_x$ , where  $m$  is a constant, typically  $m \approx 2$  near the surface of the earth. Substituting (17) and the above relationships for  $\sigma_y$  and  $\sigma_x$  into (14) we obtain upon integrating

$$\sigma_x = \frac{\sqrt{5}}{2} \beta x \quad (18)$$

or identical in form to (16) for a point source. Balancing the mass flux from the exhaust with that at point  $x$  downstream gives an average concentration in the plane cross-section (cross-wind integrated concentration) of

$$\bar{C}_x = 0.43 \frac{Q_0}{\pi(\beta^2 U x^2)} \quad (19)$$

where  $Q_0$  is in g/s. For  $\beta = 0.2$  (typical according to Wilson's measurements) we obtain

$$\bar{C}_x \approx 10.75 \frac{Q_0}{\pi U x^2} \quad (20)$$

Recasting (20) into a minimum dilution formula similar to that of Wilson's (1985) equation (12), we get

$$D_{min} = \frac{C_0}{\bar{C}_x} = \frac{U \pi x^2}{V_j A_s} = \frac{\pi x^2}{R A_s} \quad (21)$$

(where  $C_0$  is an initial concentration of exhaust contaminants) which is identical in form to (12) with the initial empirical dilution factor  $A$  set to zero.

Summarizing the essential features of roof-top calculations from flush mount sources we note:

- (1) A major assumption with regard to diffusion rate of the exhaust plume is either made or inherent in the derivation of a dilution formula and should be verified by direct measurement.
- (2) Considerable empiricism is required to offset the lack of adequately describing the physics of diffusion (which may be the only practical solution).

### 3.4.2 Concentration Calculations in the Near Wake (Region II and III of Figure 5)

For sources fully entrapped in the roof wake, most models Gifford (1960), Vincent (1977) of plume concentration assume a uniform (well mixed) concentration field in the immediate wake of the building. The wind tunnel measurements of Thompson and Lombardi (1977), and Hubert et al (1980), for cubical structures also indicate the expected dilution with increasing distance from the downstream face of their structures. Wilson (1977b) found a strong spatial distribution of concentration over a single modelled building structure. Once again, as was the case for concentrations on the roof of these structures a simple mass balance was found to give reasonable results for minimum dilution. However, the actual wind tunnel concentration data spans typically two orders of magnitude. This suggests that a great deal of work is necessary before suitable concentration predictions can be made even in a controlled laboratory experiment. Wilson and Britter (1982), describe efforts to incorporate the effect of stack height into the calculation procedures. They conclude that simple Gaussian plume theory adequately predicts concentration within a factor of 2 to 5, and that the uncertainty inherent in the complex interaction between a plume and the flow near a building simply requires one to make the stack high enough to avoid these complex recirculation regions.

An important aspect of the concentration field within the immediate building wake cavity is its temporal nature. Vincent (1977) has shown that the concentrations are associated with *residence times* in the wake cavity. Both our own visual observations of the dyed plume and concentration measurements indicate a periodic flushing action of the fluid between the upstream and downstream structures.

### 3.4.3 Concentration Calculations in the Far Wake (Region IV of Figure 5)

In the far wake region, the effects of the complex recirculation zones of the immediate wake are lost and one generally assumes that the contaminant has diffused to fill the entire building wake. Here again, recourse to modified forms of the simple Gaussian diffusion equation predominate the literature. Huber (1984) presents three models for the modified dispersion coefficients (sigmas) in the Gaussian plume model for estimating ground level concentration. The Gaussian plume equation for the normalized ground-level concentration is

$$\frac{\psi U}{Q} = \frac{\exp\left[-\frac{1}{2} \frac{H^2}{\sigma_z^2}\right]}{\pi \sigma_y \sigma_z} (\text{m}^{-2}) \quad (22)$$

where  $\psi$  = pollutant concentration ( $\text{gm}^{-3}$ )

$U$  = the mean wind speed at release height ( $\text{ms}^{-1}$ )

$Q$  = emission rate ( $\text{gs}^{-1}$ )

$H$  = plume centreline height above receptor (m)

$\sigma_y, \sigma_z$  = horizontal and vertical dispersion parameters (m).

Huber presents the following equations for enhanced dispersion parameters (all beyond 3 building heights  $H_b$ ).

Model 1:

$$\begin{aligned} \text{for } 3H_b < x < 10H_b & \left\{ \begin{array}{l} \sigma_y' = 0.7W_b/2 + 0.067(x - 3H_b) \\ \sigma_z' = 0.7H_b + 0.067(x - 3H_b) \end{array} \right. \\ \text{for } x \geq 10H_b & \left\{ \begin{array}{l} \sigma_y' = \sigma_y(x + S_y) \\ \sigma_z' = \sigma_z(x + S_z) \end{array} \right. \end{aligned}$$

Model 2:

$$\text{for } x > 3H_b \left\{ \begin{array}{l} \sigma_y' = [\sigma_y^2 + (0.7W_b/2)^2]^{1/2} \\ \sigma_z' = [\sigma_z^2 + (0.7H_b)^2]^{1/2} \end{array} \right.$$

Model 3:

$$\begin{aligned} \sigma_y' &= (\sigma_y^2 + CA/\pi)^{1/2} \\ \sigma_z' &= (\sigma_z^2 + CA/\pi)^{1/2} \end{aligned}$$

where

$x$  = distance downwind of the leeward edge of the building

$W_b/2$  = half-width of the wake

$H_b$  = height of the highest influential building

$S_y$  = a virtual source distance such that

$$\sigma_y'(10H_b) = 0.7W_b + 0.5H_b$$

$S_z$  = a virtual source distance such that

$$\sigma_z'(10H_b) = 1.2H_b$$

$\sigma_y, \sigma_z$  = horizontal and vertical dispersion parameters in absence of building influences

$A$  = cross-sectional area of buildings normal to the wind

$C$  = a constant

Once again, it is clearly evident from models for far wake concentration calculations that a significant amount of empiricism is required and built into these models. Huber's models, are essentially based on modifications to the normal undisturbed Gaussian dispersion coefficients, which themselves are empirical formulations and subject to a great deal of uncertainty within the mechanisms (stability classification schemes) that are used to select them.

### 3.5 Conclusions on Calculations of Building Wake Concentration Fields

The above brief overview of the literature on calculating building wake concentration fields indicates the extremely complex nature of this turbulent shear flow. The mathematical models are generally by necessity relatively simple. Their simplicity subsequently dictates a large empirical data base in order to be effective. At present, it would seem that a well organized effort is required to put some order into the classification of existing models. Subsequently, a thorough model validation and development study is required.

For the purposes of this report, it would seem that a case study approach to the validation and model development for building geometry and layout specific to Canadian Nuclear Power Plants is required.

The majority of all concentration calculation models pertain to sources associated with single structures. The development of concentration calculation procedures for the simple two adjacent structures (cubes) of this present study, would appear well warranted.

### 3.6 Concentration Calculations Inside Structures

At the time of writing, little work appears to have been directed to calculating the concentrations inside buildings due to sources on or near a building. Healy (1984) describes

a single calculation procedure for the interior of buildings in the path of a radioactive cloud. The calculations assume a known average concentration immediately surrounding the building (certainly not the case in reality) and that the contaminant is uniformly mixed throughout a known building volume.

The governing equation for the rate at which contaminant enters the building is determined from a straight forward mass balance and is given by

$$dq = V\bar{\psi}dt - \frac{Vq}{s}dt \quad (23)$$

where  $dq$  is a differential amount of contaminant (kg) entering the building over the differential time increment  $dt$ .  $\bar{\psi}$  is the average concentration surrounding the building,  $V$  is the ventilation rate ( $\text{m}^3 \text{s}^{-1}$ ) and  $s$  is the building volume ( $\text{m}^3$ ). Thus,  $q/s$  is the concentration of contaminant in the building at time  $t$ . Equation (23) integrates, with initial contaminant concentration  $q/s = 0$  at  $t = 0$  to:

$$\frac{q}{s} = \bar{\psi}[1 - \exp(-\frac{V}{s}t)] \quad (24)$$

where  $V/s$  represents the building air exchange rate ( $\text{s}^{-1}$ ).

Equation (24) is identical in form to that given by Truong and Asmis (1984) in their assessment report AR132.

Combinations of equation (24) may be linked together to estimate concentrations in various subsections of a building (rooms). The points of entry into a given building and corresponding ventilation rates (transport throughout the building) must be known.

The exposure dosage is obtained from the time integration of (24), which consists of the dosage obtained over the time period that the contaminant cloud is present around the building plus the subsequent decaying dosage after the cloud has passed by the building. The net result is

$$\left(\frac{q}{s}\right) = \bar{\psi} \left[ t_1 - \frac{s}{V} [1 - \exp(-\frac{V}{s}t_1)] (1 - \frac{1}{\eta}) \right] \quad (25)$$

where  $\eta$  is a ventilation rate factor such that  $\eta V$  is the ventilation rate after cloud passage. The value  $\eta$  would depend on wind direction, speed, opening of windows or turning on ventilation systems. If for example, the ventilation rate were constant ( $\eta = 1$ ), the dosage would be  $\bar{\psi}t_1$ , the same as that outside the building. If the ventilation rate decreases ( $\eta < 1$ ), then the concentration integral in the building would be greater than that outside. Thus equation (25), suggests that sealing the building envelope during cloud passage and then maximizing the ventilation rate immediately after cloud passage minimizes exposure.

Moschandreas (1985), in his review paper describes a similar equation to (25) but accounts for sources within the building.

## 4 EXPERIMENTAL DESIGN AND PROCEDURE

From the foregoing scale analysis, it would appear that in order to successfully model the dispersion of stack gases and their interaction with the wake of adjacent structures, we must first model the neutral planetary boundary layer. However, there appears to be some question as to the extent that ambient turbulence structure affects the near-field flow. The model structure should scale with the boundary layer depth and the height of the constant shear stress or surface layer.

## 4.1 Modelling the Neutral Planetary Boundary Layer in a Water Channel

Many techniques have been developed which artificially create deep turbulent boundary layers in relatively short streamwise fetches. Figure 8 below illustrates four such techniques. The long 25 m naturally developed boundary layer in the Colorado State University wind tunnel contrasts dramatically with the relatively short fetches of the artificially generated boundary layer of order 4 m.

The generated boundary layers should be two-dimensional, slowly changing in the streamwise direction and Reynolds number independent. In the present work, the Counihan (1969) approach is used to generate a neutrally stable shear flow in the Environmental Fluid Mechanics Laboratory water channel in the Department of Mechanical Engineering, University of Waterloo. The simulation system (Figure 9) consists of a castellated barrier wall, a set of quarter-elliptic, constant wedge angle ( $12^\circ$ ) vorticity generators, and an aerodynamically rough surface. The barrier wall height and surface roughness are selected for the desired flow characteristics (ie. for a rural or urban simulation).

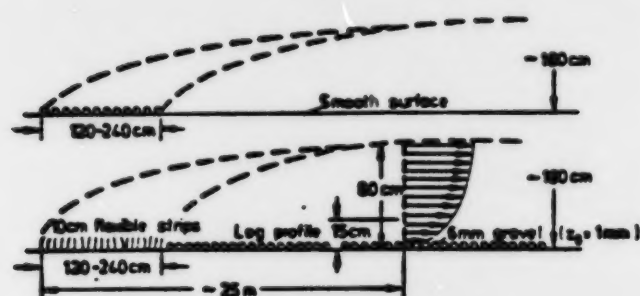
The nominal configuration of the boundary layer simulation system is set up for a rural simulation, using standard LEGO base boards as the surface roughness. The height of the vorticity generators is  $H = 40$  cm, relative to the 61 cm deep water channel flow. For this set-up the distance from the barrier wall to the test section (plume source location) is  $9H$ , shown by Robins' (1970) to be an adequate fetch for plume studies. The depth of the boundary layer in the test section is approximately  $H$ .

## 4.2 Model Cubes

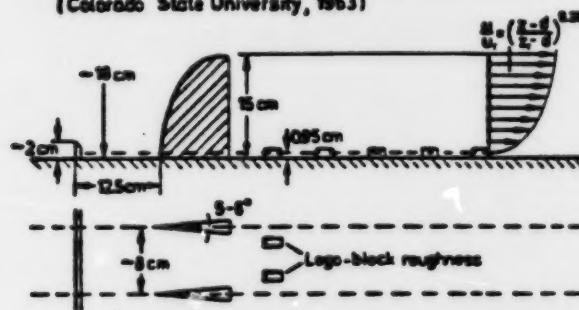
Since we estimate that our modelled planetary boundary layer height  $\delta$  will be 40 cm, and Counihan et al (1974) suggest that  $\delta/H = 8$ , we choose a model dimension of 5 cm for our cubic structures. Blockage effects with a 5 cm cube in our water channel are insignificant as the ratio of model cross-sectional area to working section cross-flow is 0.0034. If we were to model a specific site, more care would have to be taken to ensure that geometric similitude was maintained. However, this would require detailed knowledge of the site wind field, as well as site layout and geometry.

## 4.3 Model Stack Design

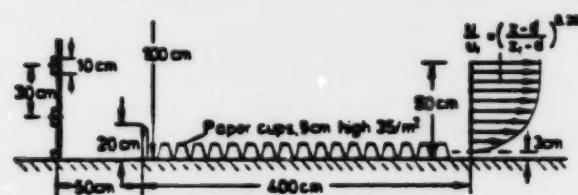
The stack for this study must only serve to act as a source of a tracer that can be measured at various points downstream of the source. We will take advantage of water as the model fluid (where its density is a modest function of temperature compared to air) and use temperature to represent a passive tracer or contaminant. Thus the stack exit velocity, and diameter are chosen to give us a detectable excess temperature ( $\Delta T/\Delta T_a$ ) at a distance of 4 cube dimensions downstream (lee of downstream cube). Since we have control over the  $\Delta T_j$ , we can determine this experimentally such that we keep  $\Delta T_j$  to a minimum, ensuring passive behavior of the fluid elements. Golden (1961) indicates that temperature may serve as a suitable passive tracer (scalar) providing the ambient Froude number does not drop below 0.8. In our experiments, we easily satisfy this criterion for very near the stack exit our Froude number is 3.75 and is proportional to the inverse of excess temperature in the plume resulting in much higher values further from the stack.



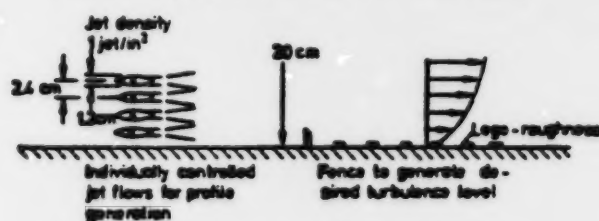
a. Boundary layer generation along test section floor (Colorado State University, 1963)



b. Boundary layer generation with vortex generators (Counihan, 1971)



c. Boundary layer generation with fence (Cook, 1973)



d. Boundary layer generation with jets (Tourassis, 1972)

Figure 8: Methods for Generating Boundary Layer Flows in a Wind Tunnel

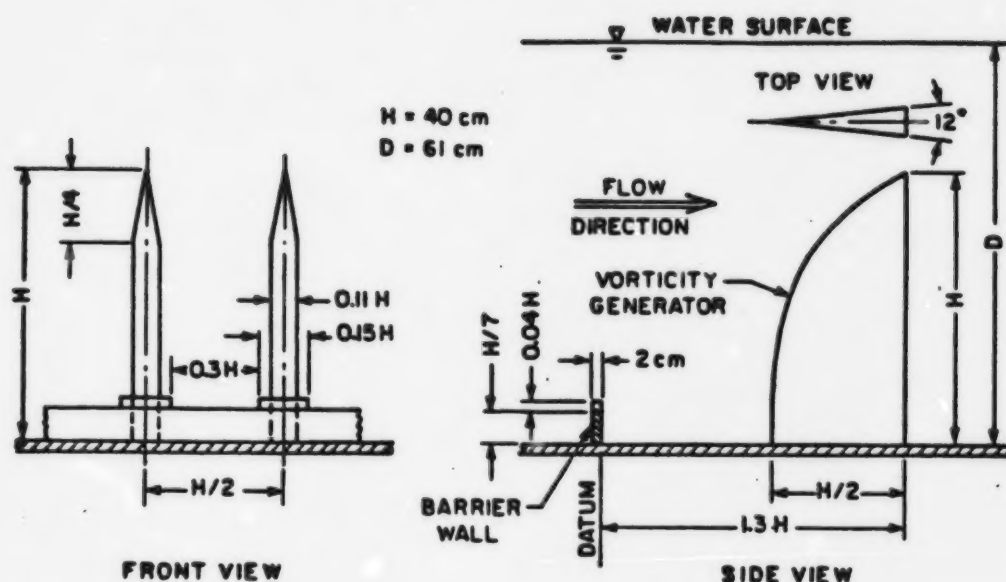


Figure 9: Arrangement of the Boundary Layer Simulation System

## 5 TEST MODEL EXPERIMENTS

The objective of the model test experiments is to determine the relationships between the effective stack height and plume building wake interaction that leads to recirculation and or interference for cube spacings  $S$  of  $1/2$ ,  $1$ ,  $2$ , and  $3$  cube dimensions. Thus, an experiment will consist of time-averaged measurements of the  $(\Delta T/\Delta T_0)$  concentration in planes normal to the approach flow direction. Table 1 describes the experimental conditions for these measurements and Figure 10 illustrates the locations at which the measurements were made for each of the above spacings.

Flow visualization experiments of the plume and wake were conducted for each of the proposed cube spacings to illustrate the nature of the plume wake interaction. The visualization experiments were recorded on fixed frame 35 mm film and on a video tape.

### 5.1 Data Reduction and Analysis

Table 2 and Figure 11 indicate the pertinent source and ambient data for the 41 individual plume cross-section measurements recorded. Table 2 defines the cube spacing and source release height relationship for each test as well as the location of each plume cross-section measurement. It is worthy to note that we made some measurements with a single cube in order to provide a reference for the double cube measurements. In order to maintain firm control over the release height, cube height, and spacing relationship, the plume emitted horizontally to the flow. Thus, the results best represent a downwashing vent plume, which at the normally low vent velocity to wind speed ratios is probably most appropriate.

### 5.2 Results and Discussion

Figures 12 to 19 illustrate the effects of plume-wake interaction for the three plume release heights of  $0D$ ,  $D/2$ , and  $D$ . Each figure represents data obtained for a single plume release

Table 1: Summary of Experimental Conditions for the Concentration Measurements

|                                                                              |                                  |
|------------------------------------------------------------------------------|----------------------------------|
| Cube dimension, $D$                                                          | $= 5.0 \text{ cm}$               |
| Cube separation distances, $S$                                               | $= D/2, 1D, 2D, \text{ and } 3D$ |
| Source release heights, $h$                                                  | $= 0D, D/2, \text{ and } 1D$     |
| Source exit diameter, $d$                                                    | $= 0.4318 \text{ cm}$            |
| Source exit area, $A_s$                                                      | $= 0.1464 \text{ cm}^2$          |
| Source exit velocity, $V_j$                                                  | $= 18.6 \text{ cm/s}$            |
| Normalized source concentration level, $(\Delta T / \Delta T_s) \times 1000$ | $= 1000$                         |
| Velocity ratio, $V_j / U_a$                                                  | $= 2.0$                          |
| Jet Froude no., $V_j / (gd(\Delta \rho / \rho_j))^{1/2}$                     | $= 18.0$                         |
| Amb. Froude no., $U_a / (gD(\Delta \rho / \rho_a))^{1/2}$                    | $= 3.75$                         |
| Measurement planes                                                           | $= A(S = 0), B(S/2), C(S + D)$   |

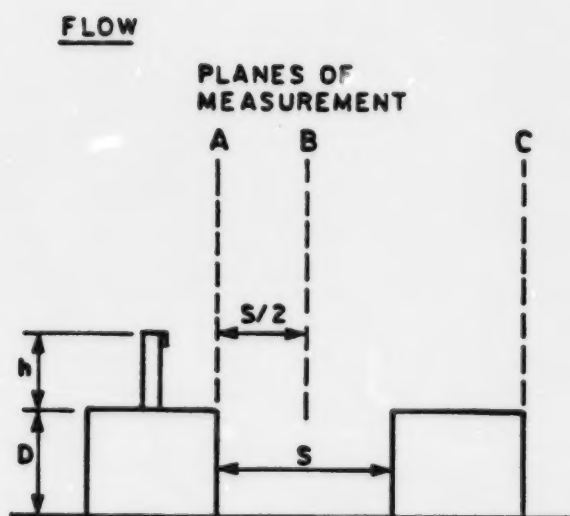


Figure 10: Schematic Diagram of the Experimental Configuration Showing the Planes of Measurement: (A) Downstream Face of the Upstream Cube, (B) Midpoint Between the Two Cubes, (C) Downstream Face of the Downstream Cube.

Table 2: Summary of Cross-sectional Concentration Measurements

| Test No. | Figure No. | Cube Spacing | Source Release Height | Cross-section Location, $x$ (cm) |
|----------|------------|--------------|-----------------------|----------------------------------|
| 1a       | 12a        | single cube  | 0D                    | A                                |
| 1b       | -          | single cube  | 0D                    | $S = 4.5D$                       |
| 2a       | 13a        | single cube  | $D/2$                 | A                                |
| 2b       | -          | single cube  | $D/2$                 | $S = 4.5D$                       |
| 3a       | -          | single cube  | 1D                    | A                                |
| 3b       | -          | single cube  | 1D                    | $S = 4.5D$                       |
| 4a       | 12b        | $D/2$        | 0D                    | A                                |
| 4b       | 17a        | $D/2$        | 0D                    | C                                |
| 5a       | 13b        | $D/2$        | $D/2$                 | A                                |
| 5b       | 18a        | $D/2$        | $D/2$                 | C                                |
| 6a       | -          | $D/2$        | 1D                    | A                                |
| 6b       | 19a        | $D/2$        | 1D                    | C                                |
| 7a       | 12c        | 1D           | 0D                    | A                                |
| 7b       | 14a        | 1D           | 0D                    | B                                |
| 7c       | 17b        | 1D           | 0D                    | C                                |
| 8a       | 13c        | 1D           | $D/2$                 | A                                |
| 8b       | 15a        | 1D           | $D/2$                 | B                                |
| 8c       | 18b        | 1D           | $D/2$                 | C                                |
| 9a       | -          | 1D           | 1D                    | A                                |
| 9b       | 16a        | 1D           | 1D                    | B                                |
| 9c       | 19b        | 1D           | 1D                    | C                                |
| 10a      | 12d        | 2D           | 0D                    | A                                |
| 10b      | 14b        | 2D           | 0D                    | B                                |
| 10c      | 17c        | 2D           | 0D                    | C                                |
| 11a      | 13d        | 2D           | $D/2$                 | A                                |
| 11b      | 15b        | 2D           | $D/2$                 | B                                |
| 11c      | 18c        | 2D           | $D/2$                 | C                                |
| 12a      | -          | 2D           | 1D                    | A                                |
| 12b      | 16b        | 2D           | 1D                    | B                                |
| 12c      | 19b        | 2D           | 1D                    | C                                |
| 13a      | 12e        | 3D           | 0D                    | A                                |
| 13b      | 14c        | 3D           | 0D                    | B                                |
| 13c      | 17d        | 3D           | 0D                    | C                                |
| 14a      | 13e        | 3D           | $D/2$                 | A                                |
| 14b      | 15c        | 3D           | $D/2$                 | B                                |
| 14c      | 18d        | 3D           | $D/2$                 | C                                |
| 15a      | -          | 3D           | 1D                    | A                                |
| 15b      | 16c        | 3D           | 1D                    | B                                |
| 15c      | 19d        | 3D           | 1D                    | C                                |
| 16a      | 20a        | 1D           | 0D <sup>1</sup>       | A                                |
| 16b      | 20b        | 1D           | 0D <sup>1</sup>       | C                                |

<sup>1</sup> denotes a vertically emitted jet

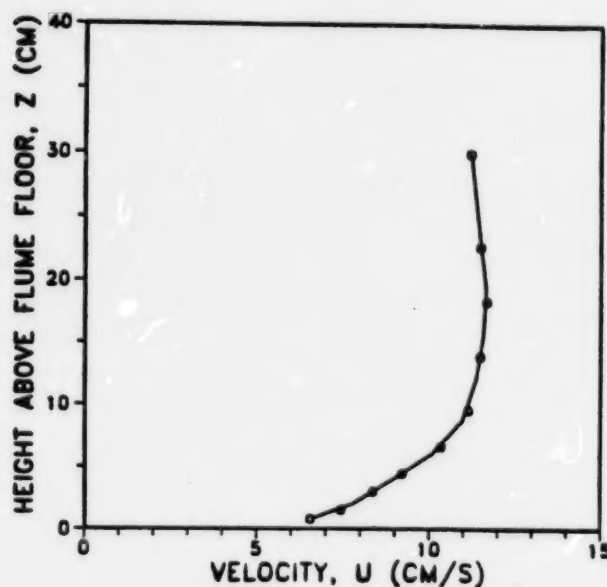


Figure 11: Water Flume Ambient Velocity Profile

height and at a single cross-section location only. Thus, each figure illustrates the effect of cube spacing on plume recirculation and or interference.

#### Recirculation at Location A:

In Figure 12, the plume release height is flush with the top surface of the cube ( $0D$ ) and the isopleths of concentration were measured at the downstream face of the upstream cube (position A of Figure 10). Figure 12(a) is for a single cube, test (1a) of Table 2, while Figures 12(b),(c),(d) and (e) indicate measured cross-sectional isopleths of concentration for  $D/2$ ,  $1D$ ,  $2D$ , and  $3D$  cube spacings respectively corresponding to test numbers (4a),(7a),(10a) and (13a) of Table 2.

Comparing Figures 12(a) and (e) we see that the cross-section concentration isopleths are essentially identical. Thus, we conclude for a flush-mount plume release (roof-top) that the downstream cube located at  $3D$  from the upwind cube has little effect on plume recirculation on the immediate downstream surface of the upwind cube.

If one focuses on the normalized concentration isopleth of 10, we see from Figure 12 that the downstream cube has a more pronounced effect on the recirculation at the downstream face of the upstream cube as the cube spacing is reduced, (ie) comparing Figure 12(e) through (b). At the  $D/2$  spacing of Figure 12(b) we see that the influence of the downstream cube shows up in the 50 normalized concentration isopleth.

Figure 12, clearly illustrates, for these test results, the relative reduction to the single cube value of plume recirculation at the downstream face of the upstream cube as cube spacing is increased from  $D/2$  to  $3D$  for a roof-top plume release.

Figure 13, illustrates the concentration isopleths measured once again, as in Figure 12,

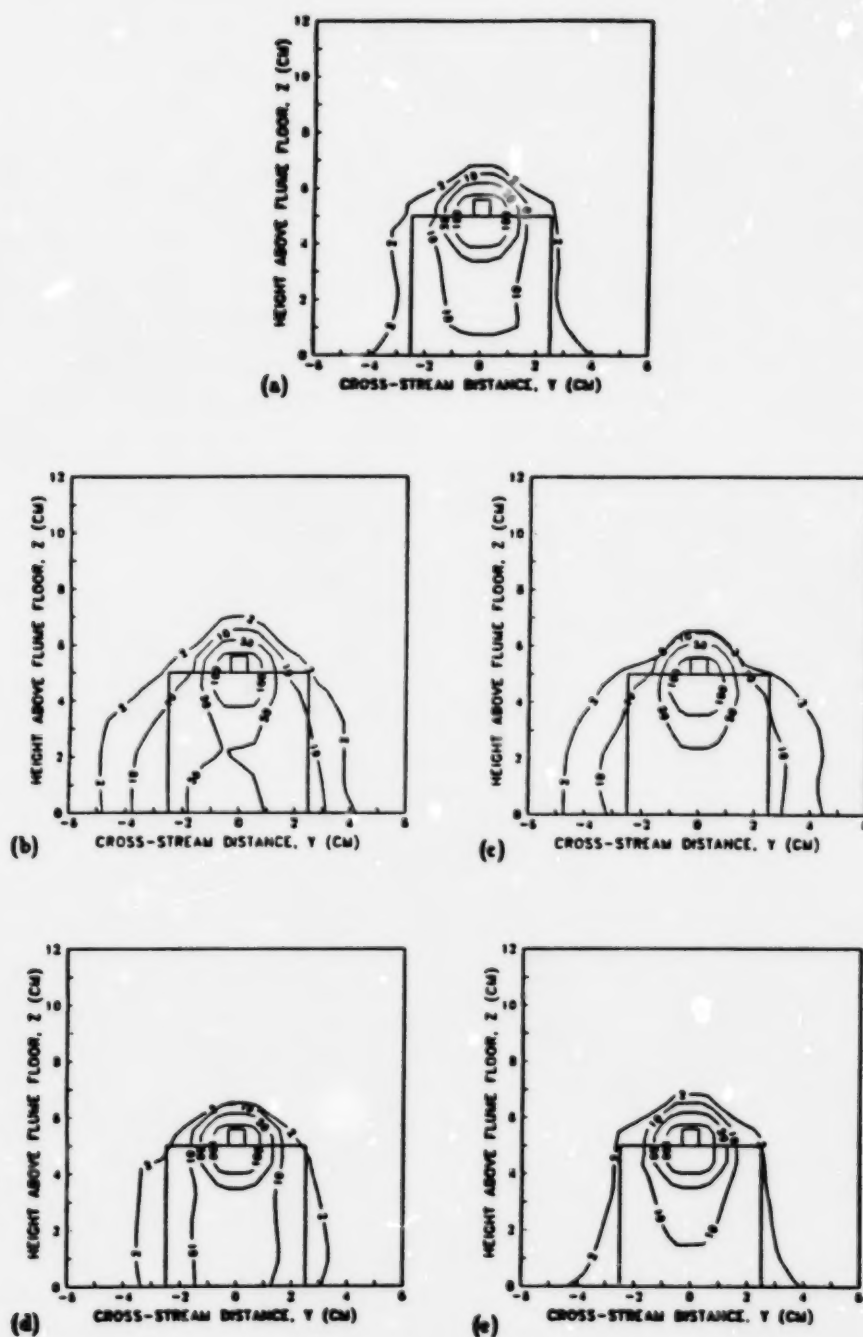


Figure 12: Cross-sectional Normalized Concentration Levels Measured (a) at Location A for a Single Cube, and (b,c,d,e) at Location A for Two Cubes In-line, Downstream from a Horizontally Emitted, Flush-Mounted Source( $h=0$ ) at Cube Spacings of (b)  $D/2$ , (c)  $1D$ , (d)  $2D$ , and (e)  $3D$ .

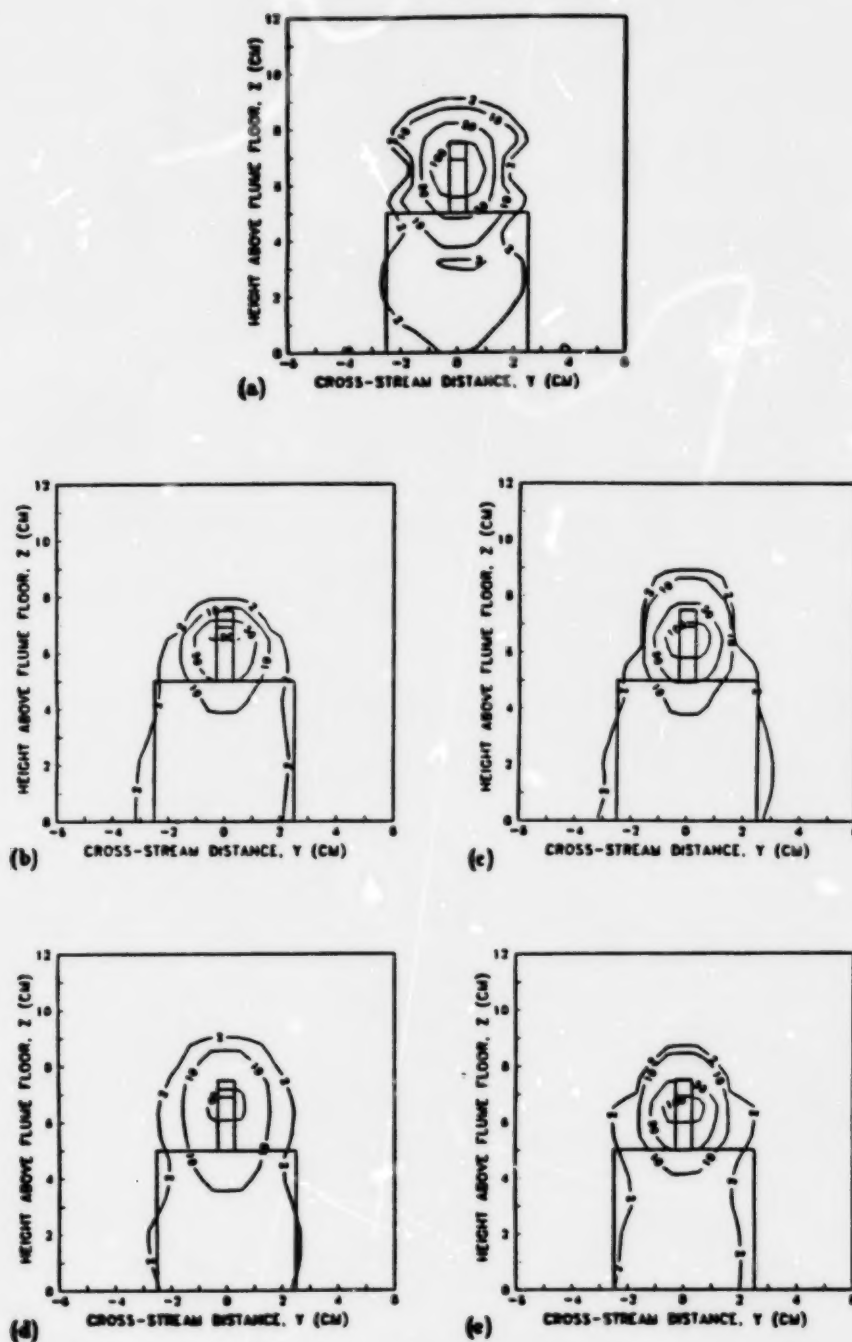


Figure 13: Cross-sectional Normalized Concentration Levels Measured (a) at Location A for a Single Cube, and (b,c,d,e) at Location A for Two Cubes In-line, Downstream from a Horizontally Emitted,  $D/2$  High Source at Cube Spacings of (b)  $D/2$ , (c)  $1D$ , (d)  $2D$ , and (e)  $3D$ .

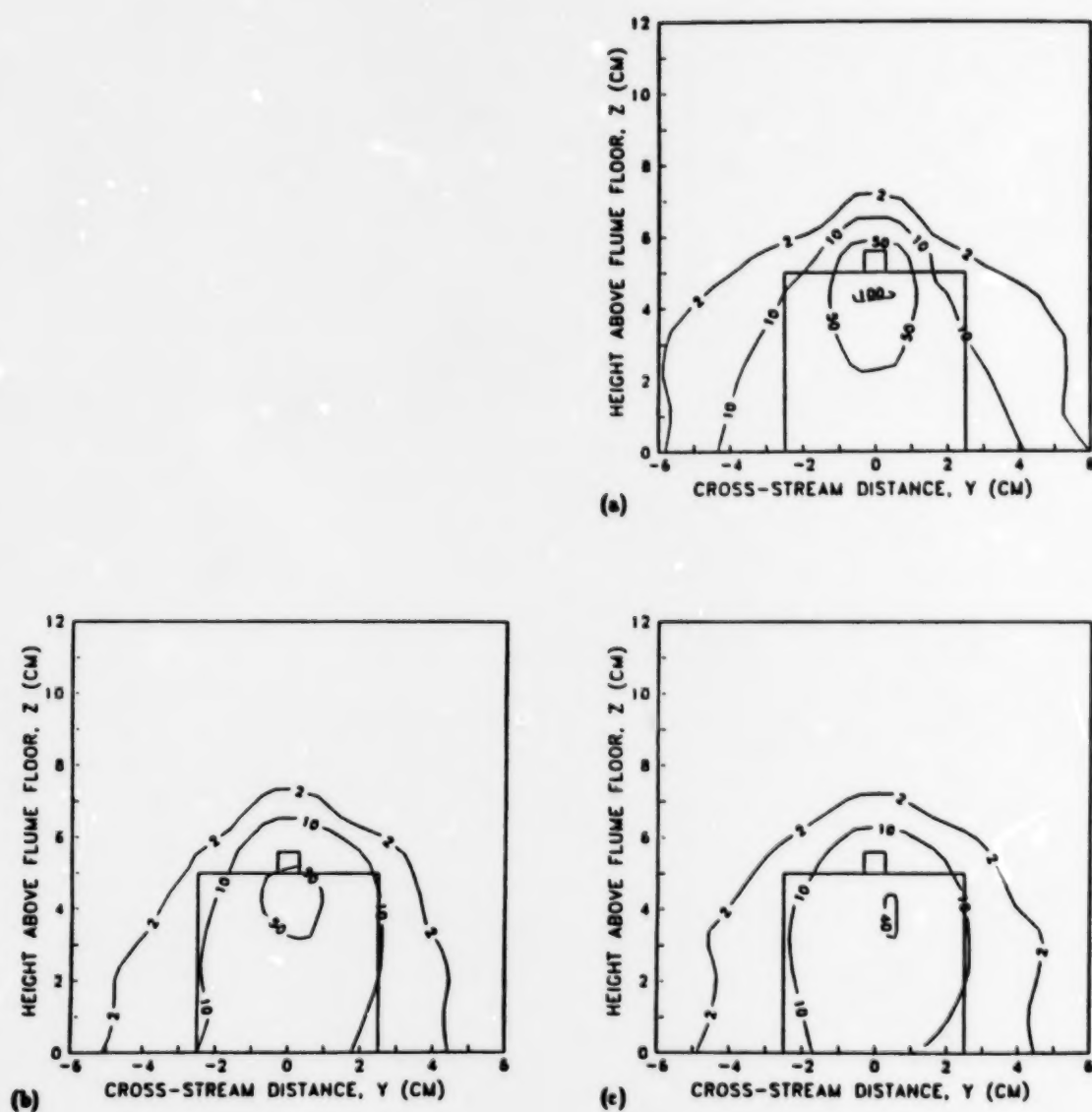


Figure 14: Cross-sectional Normalized Concentration Levels Measured at Location B, Downstream from a Horizontally Emitted, Flush-Mounted Source ( $h=0$ ) at Cube Spacings of (a)  $1D$ , (b)  $2D$ , and (c)  $3D$ .

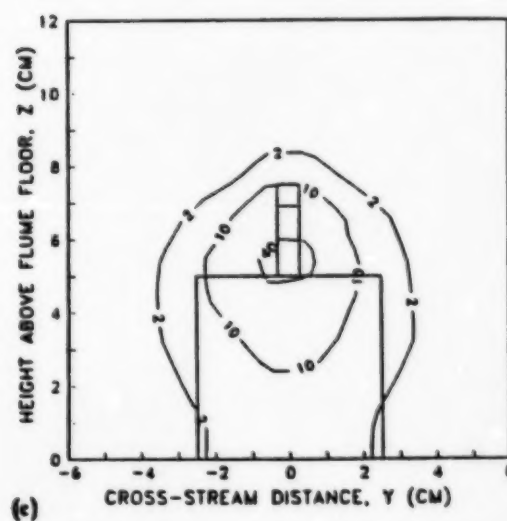
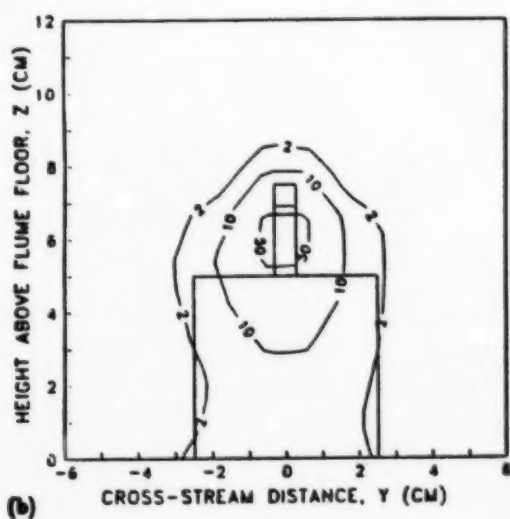
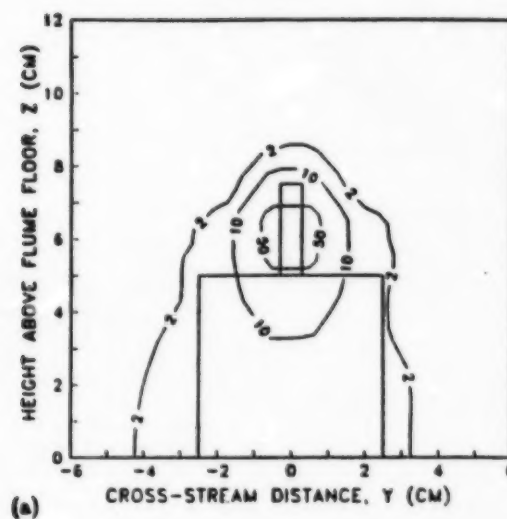


Figure 15: Cross-sectional Normalized Concentration Levels Measured at Location B, Downstream from a Horizontally Emitted,  $D/2$  High Source at Cube Spacings of (a)  $1D$ , (b)  $2D$ , and (c)  $3D$ .

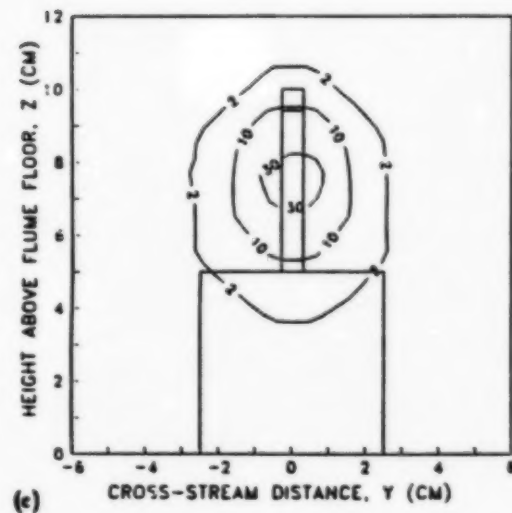
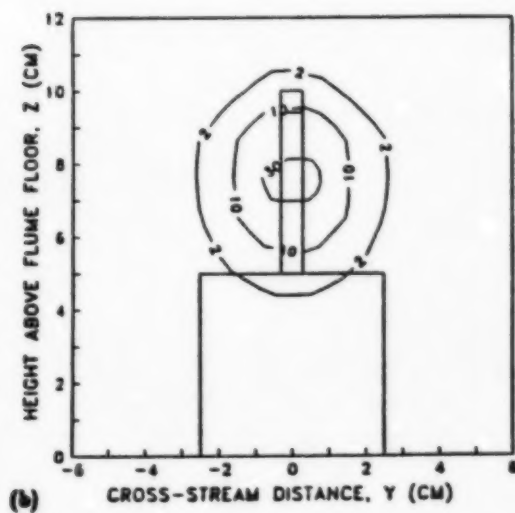
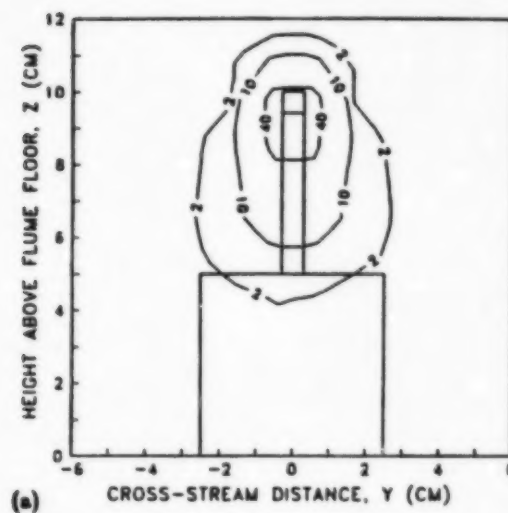


Figure 16: Cross-sectional Normalized Concentration Levels Measured at Location *B*, Downstream from a Horizontally Emitted, 1D High Source at Cube Spacings of (a)  $1D$ , (b)  $2D$ , and (c)  $3D$ .

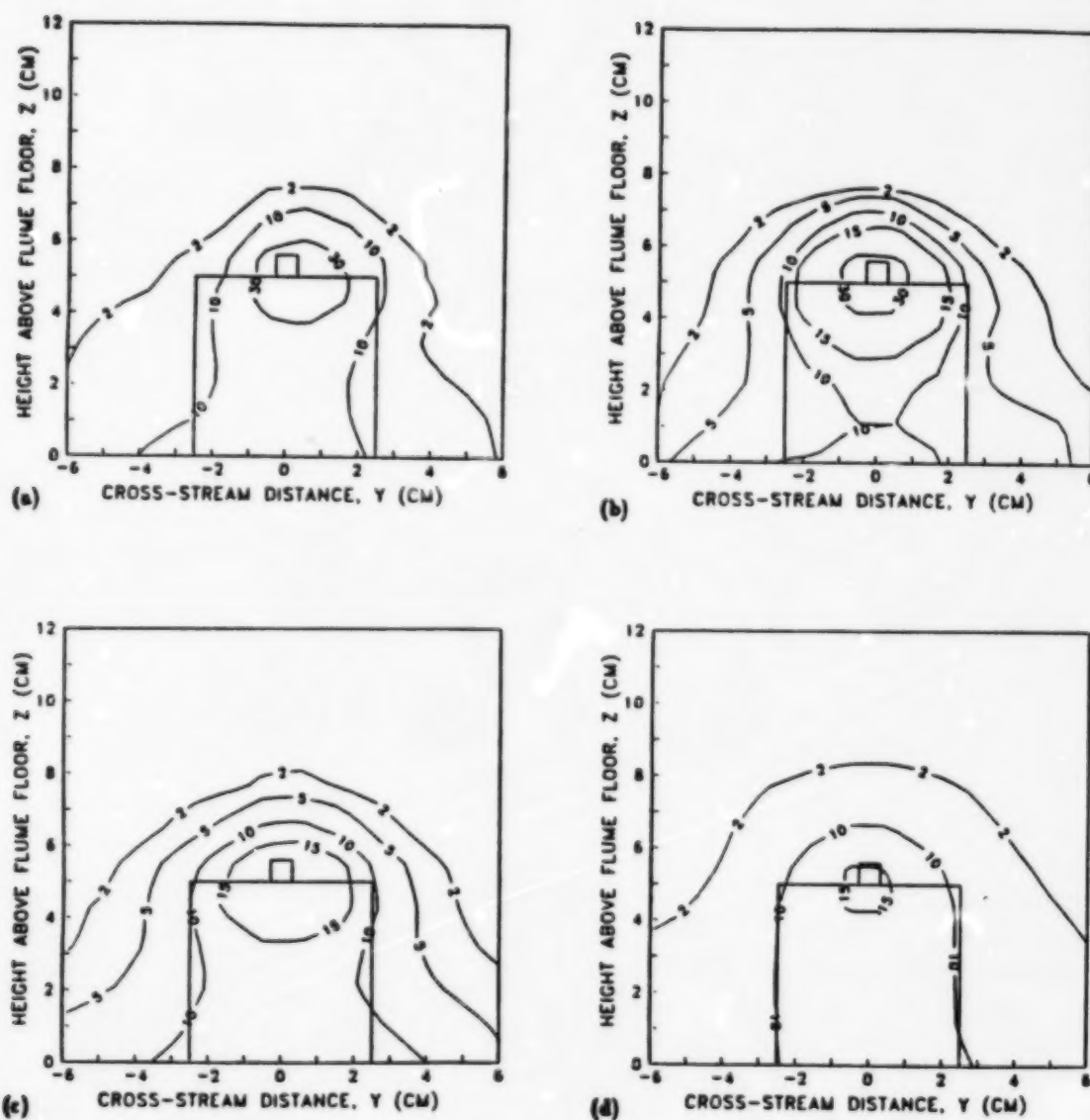


Figure 17: Cross-sectional Normalized Concentration Levels Measured at Location C, Downstream from a Horizontally Emitted, Flush-Mounted Source ( $h = 0$ ) at Cube Spacings of (a)  $D/2$ , (b)  $1D$ , (c)  $2D$ , and (d)  $3D$ .

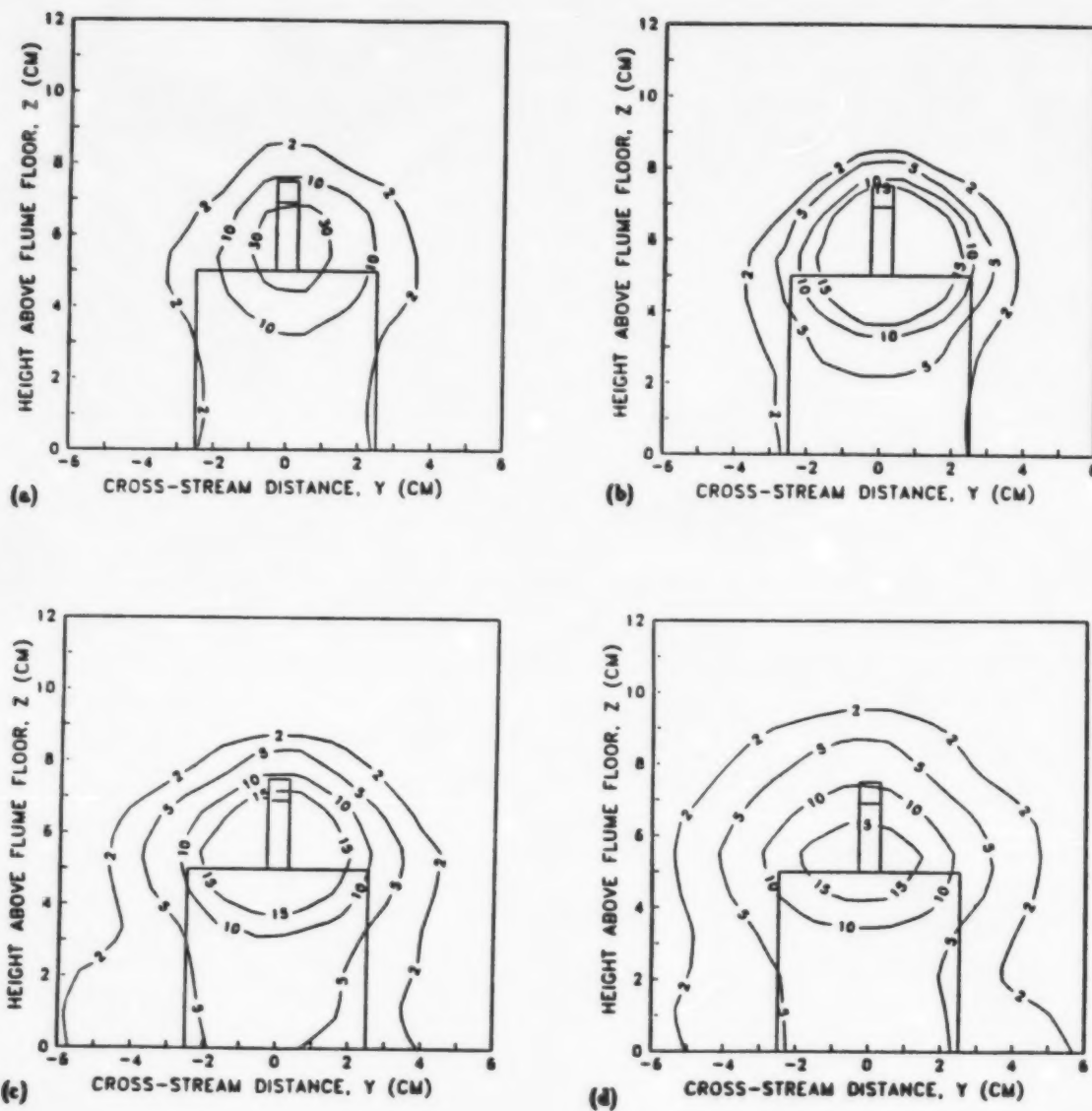


Figure 18: Cross-sectional Normalized Concentration Levels Measured at Location C, Downstream from a Horizontally Emitted,  $D/2$  High Source at Cube Spacings of (a)  $D/2$ , (b)  $1D$ , (c)  $2D$ , and (d)  $3D$ .

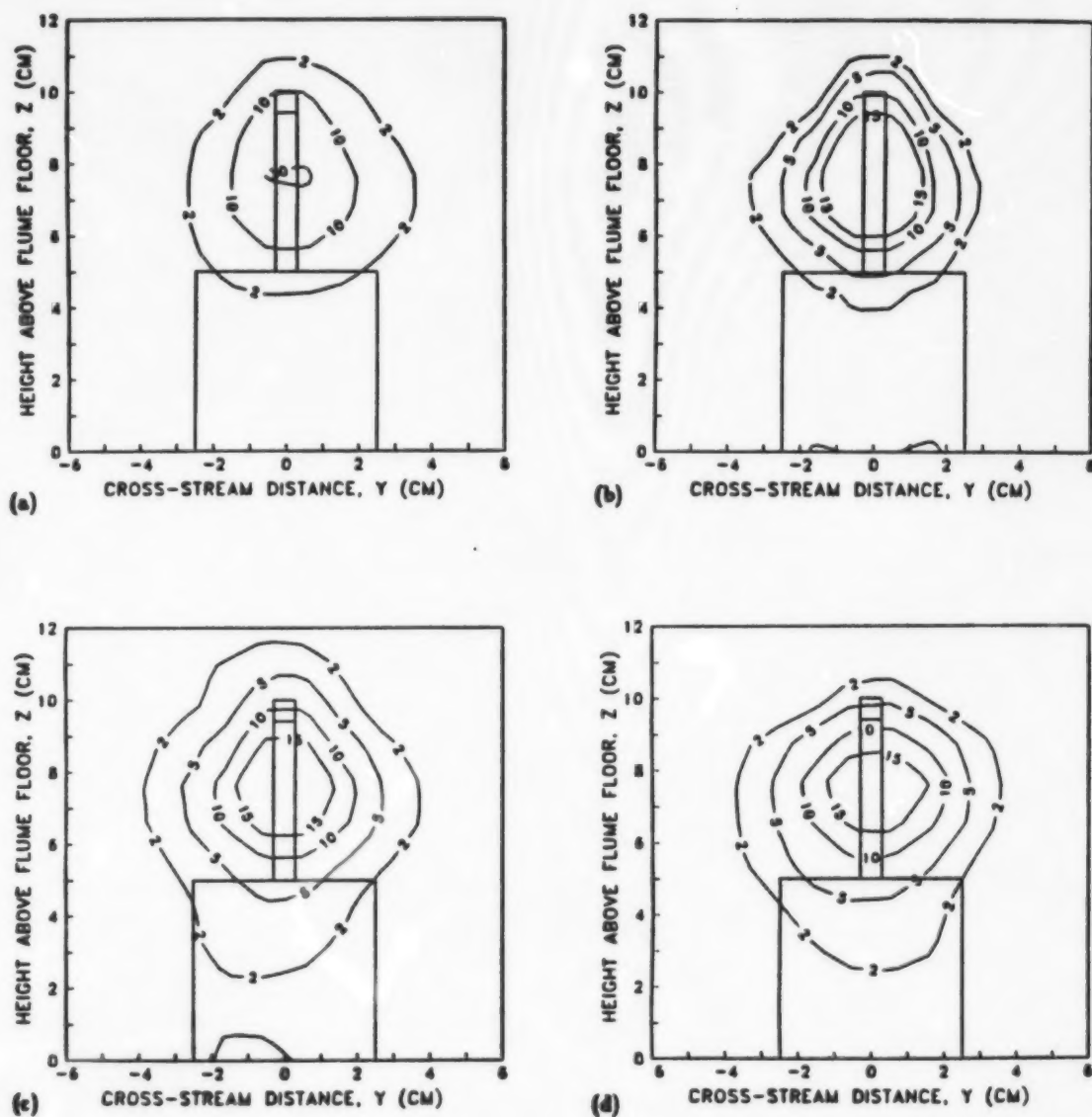


Figure 19: Cross-sectional Normalized Concentration Levels Measured at Location C, Downstream from a Horizontally Emitted, 1D High Source at Cube Spacings of (a)  $D/2$ , (b)  $1D$ , (c)  $2D$ , and (d)  $3D$ .

at the downstream face of the upstream cube (location *A*). However, the plume release height is now at  $D/2$ . Clearly, Figure 13, indicates that there is a very modest (relatively insignificant) effect of cube spacing on recirculation once we have raised our stack height to  $D/2$ . Also, the data on recirculation for a stack height of  $D$ , (not illustrated) shows as expected, even greater insensitivity to cube spacing.

#### Concluding Remarks (Recirculation at Location *A*):

For flush-mounted (roof-top) plume releases, the effect of the downstream cube on recirculation is lost at the  $3D$  cube spacing.

For plume releases at  $D/2$  or higher, there are no measureable effects on plume recirculation at cube spacings of  $D/2, D, 2D$  or  $3D$ .

#### Recirculation at Location *B*:

In Figure 14, we illustrate the plume concentration isopleths, measured at location *B* (midpoint between the two cubes except for  $D/2$  spacing) for a flush-mounted plume release. Once again, there is an obvious reduction in recirculation at the midpoint between the two cubes as cube spacing is increased and the corresponding amount of ambient fluid available for dilution is increased. We do not have a corresponding concentration isopleth measurement at  $D/2$  for the single cube for comparison purposes. However, there are only slight differences between the recirculation values shown in Figure 14 at  $2D$  and  $3D$  spacings, indicating again an insignificant influence of the downstream cube on recirculation at a  $3D$  spacing.

Figure 15, illustrates the nature of the recirculation at the midpoint between cubes for a stack release height of  $D/2$ . The principle effect of cube spacing, appears in the alteration of plume cross-sectional shape. The effect of the plume-wake interaction is reflected in this shape alteration and diminishes with increased cube spacing. The cross-sectional shape of the plume at the  $3D$  spacing is approaching that which one would expect for a single cube. The recirculation at the midpoint between cubes diminishes somewhat, but based on the results of the flush mounted plume release, we have to conclude that it is probably due to the increased distance from the source of position *B* as cube spacing is increased from  $D$  to  $3D$ .

From Figure 16, we can draw similar conclusions on the effect of cube spacing on recirculation at the midpoint location *B* for a plume release height of  $D$ .

Figures 15 and 16, both indicate that the plume centre of gravity is shifted down from stack top (plume trajectory is altered) due to the proximity of the cubes and the subsequent wake-plume interaction.

#### Recirculation at location *C*:

Similar conclusions, regarding the effect of plume spacing on the recirculation measured at the downstream face of the downstream cube (location *C* of Figure 10) may be drawn from Figures 17, 18, and 19.

The growth and hence dilution of the plume is clearly evident as location *C* moves further from the source with increased cube spacing. There is generally little difference in the overall cross-section isopleths between the  $2D$  and  $3D$  spacings for flush-mount (Figure 17) and  $D/2$  release heights (Figure 18). Thus it would appear that recirculation on the downstream face of the downstream cube (location *C*) changes little beyond a  $2D$  cube spacing.

Figure 19, illustrates somewhat the influence of the wake on the plume, in that the plume shape appears more distorted at the  $D$ , and  $2D$  spacings, than at  $D/2$  or  $3D$ .

The above Figures 12 through 19, clearly indicate the expected effect of plume release height on recirculation and or interference. The recirculation diminishes rather dramatically as release height is increased. This clearly indicates the importance of initial plume rise above the building on the intensity of recirculation and or interference.

Figure 20, simply illustrates a comparison between a flush vertically ejected plume (jet) and a horizontally ejected plume (jet) at  $D/2$  above the top of the cube. One can readily see that the results are similar, and consequently we could derive similar conclusions on the effect of cube spacing on relative recirculation for vertically ejected plumes as we have for our horizontally ejected plume source.

Figure 21 presents a series of photographs of the dyed plume from a flush-mounted, horizontally emitted source for two cubes at a  $1D$  spacing. A periodic flushing of the fluid trapped in the recirculation zone between the the upstream and downstream cubes occurs at a frequency of roughly 10 seconds. This periodic flushing action appears to be a function of cube spacing as it was strongly evident at the  $1/2D$  and  $1D$  spacings, but only weakly evident at the  $2D$  and  $3D$  spacings. One would also suspect that the frequency and strength of the flushing action, clearly evident in Figure 21, would be a function of flow Reynolds number based on cube dimensions, although presently we do not have direct evidence of this. The periodic flushing action is accompanied by a wave-like motion of the wake in plan view. This suggests that the standing vortices on the lee edge of a single cube (see Figure 1) may be the main mechanism for the flushing action. These vortices are shed alternately, similar to the Karmam street vortices which occur behind a cylinder.

### 5.3 Conclusions from Test Study

#### 5.3.1 Stack Height $0D$ (Flush-mounted Release)

For a flush-mounted (roof-top) release the plume will most likely be trapped completely within the building wake simulating that which would occur at moderate to high wind speeds. The test data indicate that recirculation and or interference although always present is reduced significantly as cube spacing is increased. The intensity of the plume recirculation on the downstream face of the upstream cube appears to be unaltered for cube spacings of  $3D$  or greater indicating that the downstream cube has little effect on the immediate wake of the upstream cube.

#### 5.3.2 Stack Height $D/2$

Raising the effective stack height to  $D/2$  reduces recirculation and interference over that found for a  $0D$  stack height. However, the plume is still significantly trapped within the wake of the cubes. With this stack height, there is little effective reduction in recirculation with increased cube spacing.

#### 5.3.3 Stack Height $D$

Here again, the recirculation is further reduced over that found at the  $0D$  and  $D/2$  stack heights due to reduced plume-wake interaction. Again, insignificant effects are observed on recirculation as cube spacing is increased.

Clearly, as one might expect, the most effective means of reducing recirculation and or interference, is to design the stack such that most of the plume escapes the immediate and or adjacent building wakes.

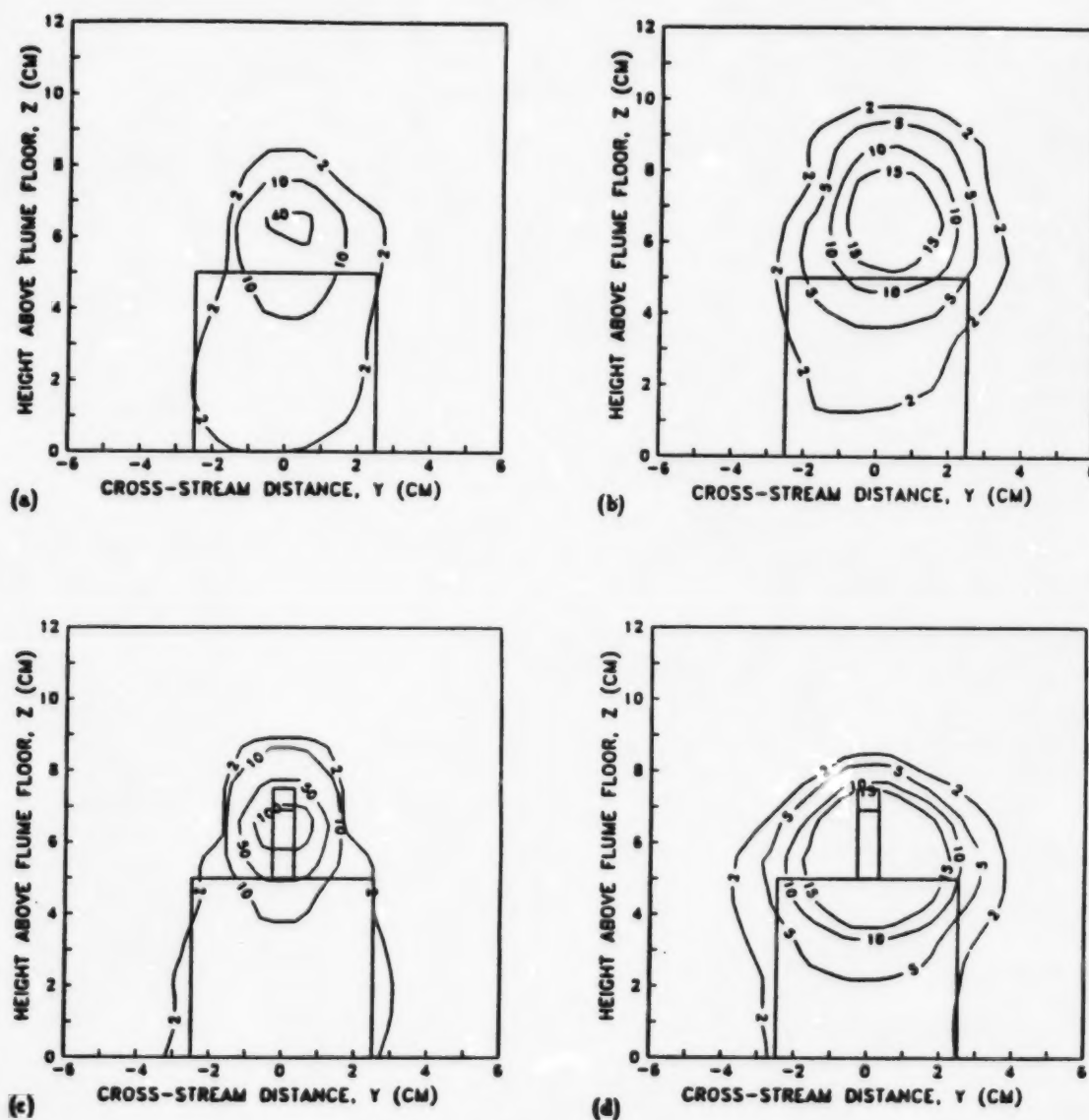


Figure 20: Cross-sectional Normalized Concentration Levels Measured for a Cube Separation Distance of  $1D$  at Downstream Locations (a) A and (b) B for a Vertically Emitted, Flush Mounted Source ( $h = 0$ ), and (c) A and (d) B for a Horizontally Emitted,  $D/2$  High Source.

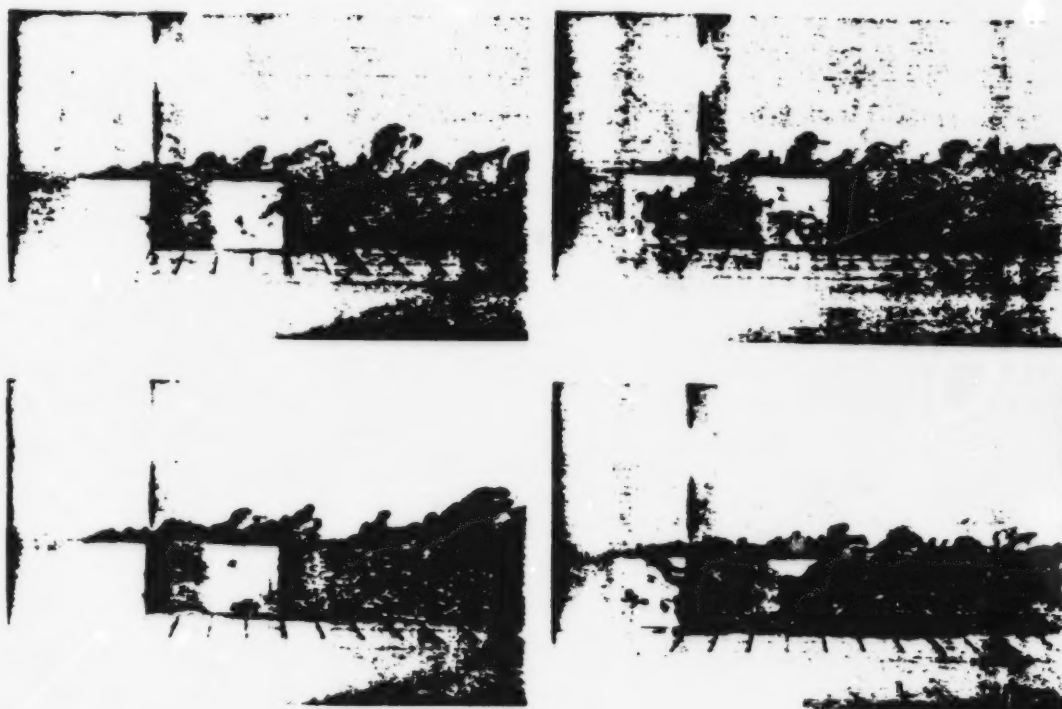


Figure 21: Photographs of the Dyed Plume from a Flush-mounted Source at a Cube Spacing of  $1D$

## 6 CONCLUSIONS

The open literature available to this study describing the nature of the complex near-field wake flows adjacent to single structures, indicates that concentration calculations in this region require a large empirical input and as such are subject to very large errors.

Comparably, little work has been done on analysing the flow about multiple structures or calculating the subsequent contaminant concentration field.

Physical modelling of specific site geometries appears to be the best engineering tool for evaluating the potential hazards of contaminant releases.

The requirements for successful physical modelling of flows about structures in the planetary boundary layer are well documented and proven.

Concentration calculations procedures for contaminant releases into the wakes of structures are generally based on modified Gaussian plume equations which restrict their generic use and contribute to large error bands when compared with specific observations.

The calculation of concentration fields within the structures themselves has received much less attention than those outside. From the literature pursued to date, there appears to be a decided lack of well documented experimental data on concentrations within structures due to sources outside the structure.

### 6.1 Model Test Study

The model test study conducted for the present work clearly indicates the relative effect that stack height and model cube separation have on recirculation and interference.

Recirculation and interference decrease with increased stack height everywhere within the concentration field.

There is an optimum separation distance for the test cubes at which the downstream cube has *no effect* on the recirculation on the downstream face of the upstream cube.

The separation distance between test cubes becomes less a factor on recirculation as stack height is increased.

Even with a stack height of one cube dimension (building height) above the cube there was enough plume-wake interaction to create significant interference (concentrations) on the downstream cube (at all cube spacings) while the upstream (source) cube recieved considerable relief from recirculation.

## 7 RECOMMENDATIONS

From the literature survey and the test model studies conducted for this report it is clear that a great deal of work is required before one could test or design a model suitable for adequately predicting concentrations within a structure due to contaminants outside the structure (particularly for multiple structures). Thus, we recommend that more realistic model tests be conducted using two in-line hollow cubic structures whereby (i) representative exhaust vents are modelled (flow rate and geometry), (ii) intake air is simulated, and (iii) interior concentrations are measured. The data from such a study would serve to test existing mathematical models or develop new ones.

In such a study, the important effects of atmospheric turbulence on the near-field concentration field could be investigated and compared with the results of the present study where insignificant background turbulence was present.

## REFERENCES

1. Blessman, J. (1971) "Pressures on Domes with Several Wind Profiles", in Proceedings of the Third International Conferences on Wind Effects on Buildings and Structures, Saikon co. Ltd., Tokyo, Sept. 6-9, 317-326.
2. Bugler, T.W., Tatinclaux, J.D. (1974) "Scale Effects on Cooling Tower Model Studies", Iowa Institute of Hydraulic Research, IIHR Report NO. 168, University of Iowa, Iowa City, Iowa, Sept (1974).
3. Castro, I.P., Robins, A.G. (1977) "The Flow Around a Surface-Mounted Cube in Uniform and Turbulent Streams", J. Fluid Mech, 79, (Pt.2), 307-335.
4. Cermak, D.J. Arya, S.P.S. (1970) "Problems of Atmospheric Shear Flow and Their Laboratory Simulation", Boundary Layer Met., 1, 40-60.
5. Cheah, S.C., Cleaver, J.W., Millward, A. (1983) "Water Channel Simulation of the Atmospheric Boundary Layer", Atmos. Environment, 17, 1439-1446.
6. Corke, T.C., Nagib, H.M., (1976) "Sensitivity of Flow Around and Pressures on a Building Model to Changes in Simulated Atmospheric Surface Layer Characteristics", IIT Fluids and Heat Transfer Report R76-1, May, Illinois Institute of Technology, Mechanics and Mechanical and Aerospace Engineering Dept., Chicago, Ill.
7. Corrsin, S. (1953) "Remarks on Turbulent Heat Transfer", Proc. First Iowa Thermo. Symp. State Univiversity of Iowa, Iowa City, 5-30.
8. Counihan, J. (1969) "An Improved Method of Simulating an Atmospheric Boundary Layer in a Wind Tunnel", Atmos. Environment, 7, 673-689.
9. Counihan, J. (1971) "An Experimental Investigation of the Wake behind a Two-dimensional Block and Behind a Cube in a Simulated Boundary Layer Flow", Central Electricity Research Laboratories, Note No RD/L/N115/71, C.E.R.L., Leatherhead, Surrey, England.
10. Counihan, J. (1975) "Adiabatic Atmospheric Boundary Layers: A Review and Analysis of Data from Period 1880-1972", Atmos. Environment, 9, 871-905.
11. Counihan, J., Hunt, J.C.R., Jackson, P.S. (1974) "Wakes Behind Two Dimensional Surface Obstacles in Turbulent Boundary Layers", J. Fluid Mech, 64, (Pt.3), 529-563.
12. Davenport, A.G., Isyumov, N. (1967) "The Application of the Boundary Layer Wind Tunnel to the Prediction of Wind Loading", In Wind Effects on Building and Structures, University of Toronto Press, Ottawa, Canada, 210-230, Sept. 1967.
13. Drivas, P.J., Shair, F.H. (1974) "Probing the Air Flow Within the Wake Downwind of a Building by Means of a Tracer Technique", Atmos. Environment, 8, 1165-1175.
12. Evans, B.H. (1957) "Natural Air Flow Around Buildings", Texas Engineering Experiment, Station Research Report No 59, Texas A&M University, College Station, Tex.
15. Frost, W., Stahabi A.M. (1977) "A Field Study of Wind over a Simulated Block Building", NASA Report CR-2804, NASA, NTIS.
16. Gifford, F.A. (1960) "Atmospheric Dispersions Calculations Using the Generalized

Gaussian Plume Model Nacl", *Safety*, 2(2), pp.56-59.

17. Golden, J. (1961) "Scale Model Techniques", M.A. Thesis, Dept. of Met. & Oceanography, N.Y. Univ.

18. Halitsky, J. (1963) "Gas Diffusion Near Buildings", ASHRAE (Am. Soc. Heating, Refrig., Air-Cond. Eng.) Trans., 69, 464-84.

19. Halitsky, J., Golden, J., Halpern, P., Wu, P. (1963a) "Wind Tunnel Tests of Gas Diffusion From a Leak in Shell of a Nuclear Power Reactor and From a Nearby Stack", Geo-phys. Sci. Lab. Report 63-2, Dept. of Meteorology & Oceanography, N.Y. University.

20. Halitsky, J. (1963b) "New York University Geophysical Science Laboratory Report", #63-3.

21. Hatcher, R.V., Meroney R.N. (1977) "Dispersion in the Wake of a Model Industrial Complex", in Preprints of Joint Conf. on Applications of Air Poll. Meteorology, Salt Lake City, Nov 29 - Dec 2, 343-346, American Met. Soc., Boston Mass.

22. Healy, J.W. (1984) "Radioactive Cloud Dose Calculations", Atmospheric Science and Power Production, D. Randerson Ed., DOE/TIC-27601, p. 685.

23. Hosker, R.P. Jr. (1979) "Empirical Estimation of Wake Cavity Size Behind Block-Type Structures", Preprints of 4th Symp. on Turbulence, Diffusion and Air Pollution, Reno, Nev., Jan 15-18, 603-609, Americ. Met. Soc., Boston Mass.

24. Hosker, R.P. (1982) "Methods for Estimating Wake Flow and Effluent Dispersion Near Simple Block-like Buildings", NRC Report NUREG/CR-252, Nuclear Regulatory Commission, NTIS.

25. Hoult, D.P. Weil, J.C. (1972) "Turbulent Plume in a Laminar Crossflow", *Atmos. Environment*, 6, 513-531.

26. Howroyd G., Slawson, P.R. (1978) "The Effects of Cross-Wind Shear on Plumes Released From Elevated Sources", A.M.S. Conference Reno Nevada Sept. 1978.

27. Huber, A.H. (1984) "Evaluation of a Method for Estimation Pollution Concentrations Downwind of Influencing Buildings", *Atmos. Environment*, 18(11), 2313-2338.

28. Huber, A.H., Snyder, W.H., Lawson, R.E. (1980) "The Effects of a Squat Building on Short Stack Effluents", U.S. Env. Prof. Agency Report EPA-600/4-80-055.

29. Hunt, J.R.C., Fernholz, H. (1975) "Wind Tunnel Simulation of the Atmospheric Boundary Layer", a report on Euronech 50, *J. Fluid Mech.* 70, 542-559.

30. Isyumov, N., Jandali, T., Davenport, A.G. (1976) "Model Studies and the Prediction of Full Scale Levels of Stack Gas Concentration", *J. Air Poll. Control Assoc.*, 26, No 10, p. 956, 1976.

31. Jain, S.C., Kennedy, J.F. (1979) "Hydrothermol Modelling of Browns Ferry Nuclear Plant Cooling Towers", IHR Report No 219, Iowa Inst. Hyd. Research, University of Iowa, Iowa City, Iowa.

32. Jain, S.C., Kennedy, J.F. (1980) "Development and Verification of Laboratory Model Techniques for Prediction of Near-Field Behaviour of Cooling-Tower Plumes", Iowa Institute of Hydraulic Research, The University of Iowa, EPRI CS-1370 RP732-1.

33. Meroney, R.N., Yang, B.T. (1970) "Gaseous Plume Diffusion about Isolated Structures of Simple Geometry", 2nd Intern. Air Poll. Conf. Proceedings, Intern. Union of Air Poll. Prevention Assoc., Dec. 6-11, 1970.
34. Meroney, R.N., Symes, C.R. (1971) "Entrainment of Stack Gases by Buildings of Rounded Geometry", Proceedings of the Air Pollution Control Association, Raleigh, N.C. April 1971, 132-135.
35. Moschandreas, D.J. (1985) "Characterization of Indoor Air Pollution", J. of Wind Engineering and Industrial Aerodynamics, 21, 39-49.
36. Norman, R.S. (1972) "On Obstacle Generated Secondary Flows in Laminar Boundary Layers and Transition to Turbulence", Phd Thesis, Illinois Institute of Technology Mechanical Department, Chicago Ill.
37. Panofsky, H.A., Dutton, J.A. (1984) "Atmospheric Turbulence", J. Wiley & Sons. New York.
38. Pasquill, F. (1974) "Atmospheric Diffusion", 2nd edition, Halsted Press, 64-65.
39. Pasquill, F. (1962) "Some Observed Properties of Medium Scale Diffusion in the Atmosphere", Quart J. Roy Soc, 88, 70-79.
40. Riley, J.J., Delisi, D.P. (1977) "Survey of Laboratory Modelling of Plume Dynamics", EPRI EA23 EPRE, Palo Alto, California.
41. Robins, A.G. (1979) "The Development and Structure of Simulated Neutrally Stable Atmospheric Boundary Layers", J. Ind. Aerodynamics 4, 71-100.
42. Saffman, P.G. (1962) "The Effect of Wind Shear on the Horizontal Spread from an Instantaneous Ground Source", Quart. J. Roy Soc, 88, 382-393.
43. Sagendorf, J.F., Ricks, N.R., Start G.E., Dickson C.R. (1979) "Near Building Diffusion Determined from Atmospheric Tracer Experiments", Proc. Fourth Symposium on Turbulence, Diffusion and Air Pollution, American Met. Soc., Reno, Nevada, Jan 1979.
44. Slawson, P.R. "Diffusion in the Wake of a Building", Unpublished Report, 1972.
45. Slawson, P.R., Champion, E.R., Goodman, C.H., "Field Study of Mechanical Draft Cooling Tower Plume Behaviour", Proc. Waste Heat Management and Utilization Conference Miami Beach, Florida, May 9 - 11, p.10, 1977.
46. Slawson, P.R., Csanady G.T. (1967) "On the Mean Path of Buoyant Bent-Over Chimney Plumes", J. Fluid Mech, 28, 311-322.
47. Smith, E.G. (1951) "Feasibility of Using Models for Predetermining Natural Ventilation", Res. Rep, Tex. Eng. Stn., 26.
48. Snyder, W.H. (1972) "Similarity Criteria for Application of Fluid Models to Study of Air Poll. Met.", Boundary - Layer Met. 3, 113-134.
49. Thompson, R.S., Lombardi, D.J., (1977) "Dispersion of Roof-Top Emissions from Isolated Buildings, a Wind Tunnel Study", U.S. Env. Prof. Agency Report EPA-600/4-77-006.
50. Townsend, A.A. (1956) "The Structure of Turbulent Shear Flow", Cambridge Univer-

sity Press, 315.

51. Truong, L., Asmis, K. (1984) "Assessment Report". A R132.
52. Ungate, C.D., Harleman, D.R.F., Jirka, G.H. (1975) "Mixing of Submerged Turbulent Jets at Low Reynolds Number", Report No 197, Ralph M Parsons Laboratory for Water Resources and Hydrodynamics, M.I.T., Cambridge Mass.
53. Vincent, J.H. (1977) "Model Experiments on the Nature of Air Pollution Transport Near Buildings", *Atmos. Environ.*, 11(8), 765-774.
54. Vincent, J.H. (1978) "Scalar Transport in the Near Aerodynamic Wake of Surface Mounted Cubes", *Atmos. Environment*, 12(6/7) 1319-1322.
55. Wilson, D.J. (1977b) "Dilution of Exhaust Gases from Building Surface Vents", *ASHRAE Trans.* 83, 168-176.
56. Wilson, D.J. (1979) "Flow Patterns over Flat-Roofed Buildings and Application to Exhaust Stack Design", *ASHRAE Trans.*, 85(Pt.2), 284-295.
57. Wilson, D.J. (1985) "Ventilation Intake Air Contamination by Nearby Exhausts", A.P.C.A. Specialty Conference on Indoor Air Quality in Cold Climates, Hazards and Abatement Measures, Ottawa, Ontario.
58. Wilson, D.J., Butler (1982) "Estimates of Building Surface Concentrations form Nearby Point Sources", *Atmos. Environ.*, 16(11) 2631-2642.
59. Wilson, D.J., Netterville D.J. (1978) "Interaction of a Roof-level Plume with a Downwind Building", *Atmos. Environ.* 12(5) 1051-1059.